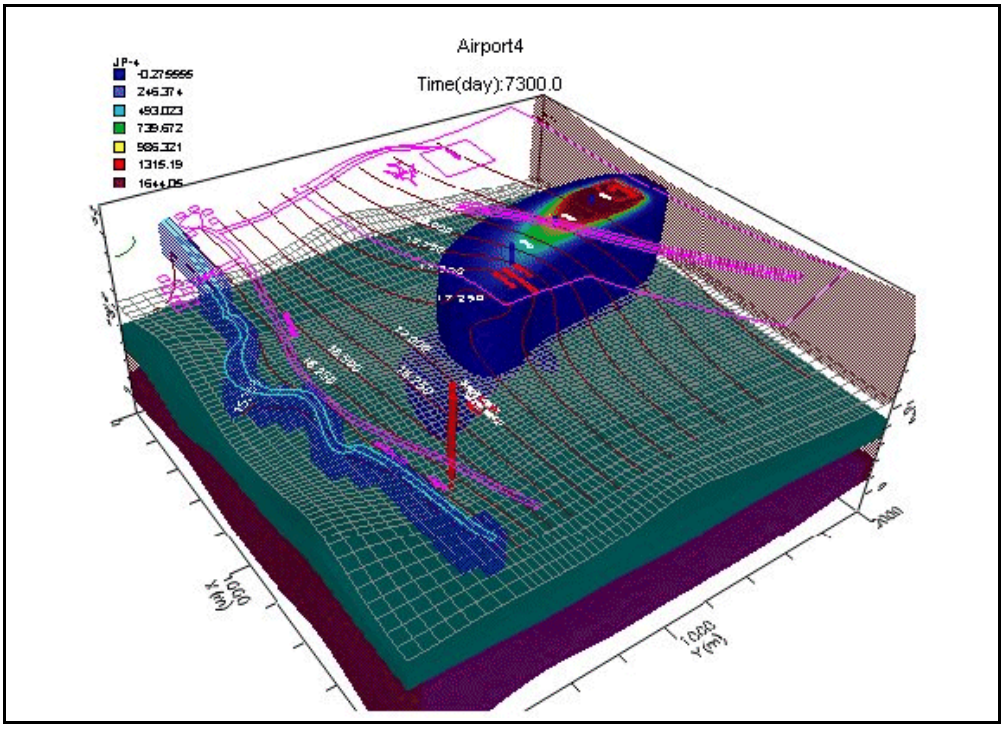
Visual MODFLOW Premium

Demo Tutorial

Includes New Features of Visual MODFLOW and a Step-by-Step Tutorial



© Schlumberger Water Services

Table of Contents

1. Introduction to Visual MODFLOW 1

What's New in Visual MODFLOW 1
New Features in Visual MODFLOW 2010.1 2
Starting Visual MODFLOW 3
Getting Around In Visual MODFLOW 3
Screen Layout 4
Contacting SWS 5

2. Visual MODFLOW Premium Tutorial 7

Introduction 7
How to Use this Tutorial 7
Terms and Notations 7
Description of the Example Model 8

3. Module I: Creating and Defining a Flow Model 9

Section 1: Generating a New Model 9
Step 1: Project Settings 10
Step 2: Flow Settings 11
Step 3: Transport Settings 12
Step 4: Model Domain 13
Add Air Photo 16
Section 2: Refining the Model Grid 19
Section 3: Adding Pumping Wells 27
Section 4: Assigning Flow Properties 30
Section 5: Assigning Flow Boundary Conditions 34
Assigning Aquifer Recharge to the Model 34
Assigning Constant Head Boundary Conditions 35
Assigning River Boundary Conditions 38
Section 6: Particle Tracking 40

4. Module II: The Contaminant Transport Model 41

Section 7: Setting Up the Transport Model 41
Contaminant Transport Properties 41
Viewing Sorption Properties 41
Assigning Dispersion Coefficients 42
Section 8: Transport Boundary Conditions 43
Section 9: Adding Observation Wells 43

5. Module III: Running Visual MODFLOW 45

Section 10: MODFLOW Run Options 45
MODFLOW Solver Options 46
Dry Cell Wetting Options 46
Section 11: MT3DMS Run Options 47
MT3DMS Solver Settings 47
MT3DMS Output Times 48
Section 12: Running the Simulation 48

6. MODULE IV: Output Visualization 49

Section 13: Head Contours 50
Displaying a Color Map of Heads 52
Section 14: Velocity Vectors 53
Section 15: Pathlines 55
Section 16: Concentration Contours 57
Section 17: Concentration vs. Time Graphs 62

7. Module V: Visual MODFLOW 3D-Explorer 63

Section 18: Starting Visual MODFLOW 3D-Explorer 64
Section 19: Changing the Default Settings 65
Section 20: Using the Navigation Tools 65
Section 21: Creating a 3D Contaminant Plume 66
Customizing the Color Legend 68
Selecting Output Times 69
Section 22: Visualizing Model Input Data 69
Section 23: Defining Surfaces (Cross-Sections and Slices) 70
Section 24: Creating a Cut-Away View 72
Section 25: Creating a Color Map of Head Values 74
Customizing the Color Legend 75
Section 26: Printing to a Graphics File 76

1

Introduction to Visual MODFLOW

Visual MODFLOW is the most complete and easy-to-use modeling environment for practical applications in three-dimensional groundwater flow and contaminant transport simulations. This fully-integrated package combines MODFLOW, MODFLOW-SURFACT, MODPATH, ZoneBudget, MT3Dxx/RT3D, MGO, and WinPEST with the most intuitive and powerful graphical interface available. The logical menu structure and easy-to-use graphical tools allow you to:

- easily dimension the model domain and select units
- conveniently assign model properties and boundary conditions
- run the model simulations
- calibrate the model using manual or automated techniques
- optimize the pumping well rates and locations
- visualize the results using 2D or 3D graphics

The model input parameters and results can be visualized in 2D (cross-section and plan view) or 3D at any time during the development of the model, or the displaying of the results. For complete three-dimensional groundwater flow and contaminant transport modeling, Visual MODFLOW is the only software package you will need.

When you purchase Visual MODFLOW, or any Schlumberger Water Services (SWS) software product, you not only get the best software in the industry, you also gain the reputation of the company behind the product, and professional technical support for the software from our team of qualified groundwater modeling professionals. SWS has been developing groundwater software since 1989, and our software is recognized, accepted and used by more than 10,000 groundwater professionals in over 90 countries around the world. This type of recognition is invaluable in establishing the credibility of your modeling software to clients and regulatory agencies. Visual MODFLOW is continuously being upgraded with new features to fulfill our clients needs.

What’s New in Visual MODFLOW

The main interface for Visual MODFLOW has much of the same user-friendly look and feel as the previous versions of Visual MODFLOW, but what’s “under the hood” has been dramatically improved to give you more powerful tools for entering, modifying, analyzing, and presenting your groundwater modeling data. Some of the more significant upgrade features in the latest version of Visual MODFLOW are described below.

New Features in Visual MODFLOW 2010.1

Support for Lake Package (LAK3)

Simulate Lake-Aquifer interactions in transient or steady-state flow conditions using the MODFLOW Lake Package (LAK3). The LAK3 package is supported in the following engines: MODFLOW 2000, MODFLOW 2005, SEAWAT and MODFLOW-SURFACT.

For more information on the LAK3 package, please refer to the following USGS publication: *Documentation of a Computer Program to Simulate Lake-Aquifer Interaction Using the MODFLOW Ground-Water Flow Model and the MOC3D Solute-Transport Model.*

Support for the Evapotranspiration Segments Package (ETS1)

Simulate evapotranspiration by defining a multi-segment function that relates evapotranspiration rate to hydraulic head. This capability provides a degree of flexibility not supported by the original EVT Package, which by contrast, simulates evapotranspiration with a single linear function.

Import MODFLOW Packages into Existing Projects

MODFLOW packages can now be directly imported into any existing Visual MODFLOW project.

Supported packages include: HFB1, HFB6, DRN, CHD, RIV, GHB, LAK, WEL and Zone Budget

More Options for Importing and Exporting Data

Visual MODFLOW now provides more flexibility for importing and exporting data to/ from your Visual MODFLOW project. The following options are now available:

- Import and export grid and property zones in layer, row and column format (IJK)
- Import and export boundary conditions in layer, row and column format for the following packages: Constant Head (CHD), Drain (DRN), River (RIV), General Head (GHB), Wall (HFB) and Lake (LAK).

Support for Windows 7

Visual MODFLOW is now supported on Windows 7 operating systems, including Professional, Enterprise and Ultimate editions.

About the Interface

The Visual MODFLOW interface has been specifically designed to increase modeling productivity, and decrease the complexities typically associated with building three-dimensional groundwater flow and contaminant transport models. In order to simplify the graphical interface, Visual MODFLOW is divided into three separate modules:

- The **Input** section
- The **Run** section
- The **Output** section.

Each of these modules is accessed from the **Main Menu** when a Visual MODFLOW project is started, or an existing project is opened. The **Main Menu** serves as a link to

seamlessly switch between each of these modules, to define or modify the model input parameters, run the simulations, calibrate and optimize the model, and display results.

The **Input** section allows the user to graphically assign all of the necessary input parameters for building a three-dimensional groundwater flow and contaminant transport model. The input menus represent the basic "model building blocks" for assembling a data set using MODFLOW, MODFLOW-SURFACT, MODPATH, ZoneBudget, and MT3D/RT3D. These menus are displayed in a logical order to guide the modeler through the steps necessary to design a groundwater flow and contaminant transport model.

The **Run** section allows the user to modify the various MODFLOW, MODFLOW-SURFACT, MODPATH, MT3D/RT3D, MGO, and WinPEST parameters and options which are run-specific. These include selecting initial head estimates, setting solver parameters, activating the re-wetting package, specifying the output controls, etc. Each of these menu selections has default settings, which are capable of running most simulations.

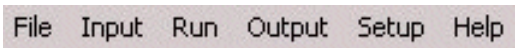
The **Output** section allows the user to display all of the modeling and calibration results for MODFLOW, MODFLOW-SURFACT, MODPATH, ZoneBudget, and MT3D/RT3D. The output menus allow you to select, customize, overlay the various display options, and export images for presenting the modeling results.

Starting Visual MODFLOW

Once Visual MODFLOW has been installed on your computer, simply double-click on the Visual MODFLOW shortcut icon, or click on **Start/Programs/SWS Software/Visual MODFLOW /Visual MODFLOW**.

Getting Around In Visual MODFLOW

The **Main Menu** screen contains the following options:



File	Select a file utility, or exit Visual MODFLOW.
Input	Design, modify, and visualize the model grid and input parameters.
Run	Enter or modify the model run settings and run the numerical simulations, in either project mode or batch mode.
Output	Visualize the model output (simulation results) using contour maps, color maps, velocity vectors, pathlines, time-series graphs, scatter plots, etc.
Setup	Select the “numeric engine” for both the flow and transport simulation(s).
Help	Get general information on how to use Visual MODFLOW.

A Microsoft compatible mouse performs as follows:

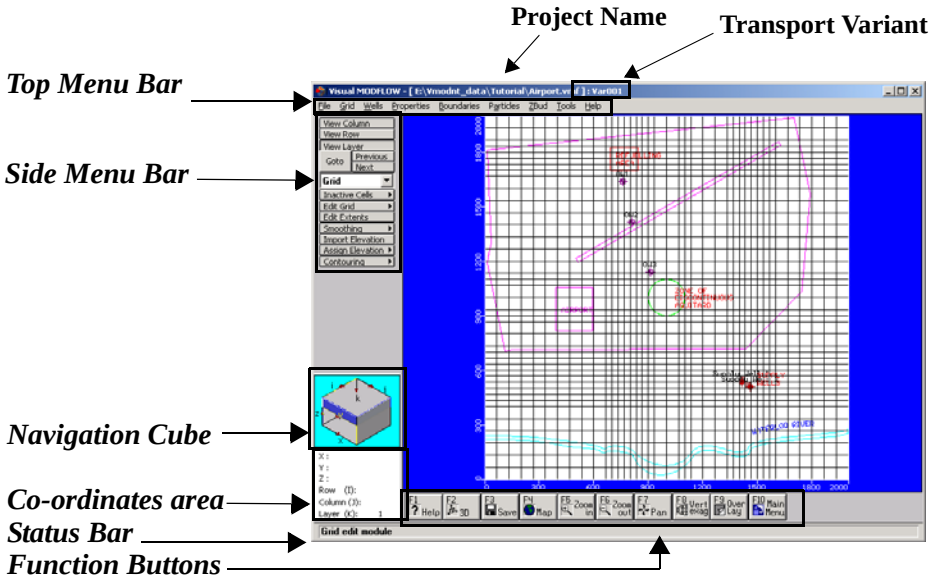
Left button: This is the regular ‘click’ button. By holding it down on top of an input box and dragging the mouse, it will also highlight the characters that will be overwritten by the next typed character.

Right button: This button has different functions depending on the context. For example, it closes polygons or ends a line during the assignment of properties, boundaries, etc. During grid design, it also locates grid rows or columns on precise co-ordinates. Using the right button brings up a sub-menu to copy, paste, delete, etc.

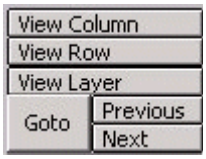
The **<Alt>** shortcut keys can be used to access toolbar items and to select many buttons and can be used anytime an item or button text has an underlined letter. Simply press the **<Alt>** key and the letter key at the same time to access the item or select the button. The number and letter keys are active only when numerical or text input is required; all other keys are ignored by Visual MODFLOW.

Screen Layout

After opening a Visual MODFLOW project and selecting the **Input** module, a screen layout similar to the figure below will appear.



Top Menu Bar: Contains specific data category menus for each section of the interface (**Input**, **Run** and **Output**).



Side Menu Bar: Contains the **View** Control buttons plus tool buttons specific for each data category. The view options are as follows:

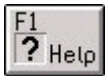
[View Column]	View a cross-section along a column.
[View Row]	View a cross-section along a row.
[View Layer]	Switch from cross-section to plan view.
[Goto]	View a specified column, row, or layer.
[Previous]	View previous column, row, or layer.
[Next]	View next column, row, or layer.

Navigator Cube: Provides a simplified 3D representation of the model domain highlighting the current layer, column, and row being viewed.

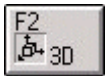
Co-ordinates Area: Shows the current location of the mouse pointer in model co-ordinates (X, Y, Z) and cell indices (I, J, K).

Status Bar: Describes the functionality and use of the feature currently highlighted by the cursor.

Function Buttons:



[F1-Help] Opens the general help window



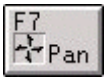
[F2-3D] Opens the Visual MODFLOW 3D-Explorer

[F3-Save] Saves the current data for the model.

[F4-Map] Import and display .DXF, .SHP, or .graphics image files (.BMP, .JPG, .TIF, .GIF, .PNG)

[F5–Zoom in] Enlarge the display of a selected rectangular area

[F6–Zoom out] Resets the display area to show the entire model domain



[F7-Pan] Drag the current view of the model in any direction to display a different model region

[F8–Vert exag] Set the vertical exaggeration value for viewing cross sections along a row or column

[F9-Overlay] Opens the Overlay Control window to make selected overlays visible and to customize the priority (order) in which they appear

[F10–Main Menu] Returns to the **Main Menu** screen.

Contacting SWS

The purpose of this tutorial is to demonstrate the ease of building, calibrating, and running a three-dimensional groundwater flow and contaminant transport model. Unfortunately, this tutorial can only cover a few of the many analytical capabilities and graphical features that make Visual MODFLOW the leading software package for groundwater flow and contaminant transport modeling.

If this tutorial has not covered some aspects of the software that you are interested in, we encourage you to continue to consult the on-line help files and the carefully read the Visual MODFLOW User's Manual. If you would like to speak with a technical representative about how Visual MODFLOW could be applied to solve your particular needs, please contact at:

Schlumberger Water Services
460 Phillip Street, Suite 101
Waterloo, Ontario, N2L 5J2

Tel: (519) 746-1798
Fax: (519) 885-5262
E-mail: sws-support@slb.com
Web: www.swstechnology.com

2

Visual MODFLOW Premium Tutorial

Introduction

This chapter contains a step-by-step tutorial to guide you through the process of using SWS' Visual MODFLOW. This tutorial will guide you through the steps required to:

- Design the model grid and assign properties and boundary conditions,
- Configure Visual MODFLOW to run the groundwater flow, particle tracking, and mass transport simulations, and
- Visualize the results using two- and three-dimensional graphics

For your convenience, Visual MODFLOW installs a complete set of input data files and simulation results for the example model that you will build. The instructions for the tutorial are provided in a step-wise format that allows you to choose the features that you are interested in examining without having to complete the entire exercise.

How to Use this Tutorial

This tutorial is divided into five modules, and each module contains a number of sections. The tutorial is designed so that the user can begin at any of the modules in order to examine specific aspects of the Visual MODFLOW environment. Each section is written in an easy-to-use step-wise format. The modules are arranged as follows.

Module I - Creating and Defining a Flow Model

Module II - The Contaminant Transport Model

Module III - Running Visual MODFLOW

Module IV - Output Visualization

Module V - Visual MODFLOW 3D Explorer

Terms and Notations

For the purposes of this tutorial, the following terms and notations will be used:

Type:- type in the given word or value

Select:- click the left mouse button where indicated

⇔ - press the **<Tab>** key

↵ - press the **<Enter>** key

☞ - **Click** the left mouse button where indicated

☞☞ - double-click the left mouse button where indicated

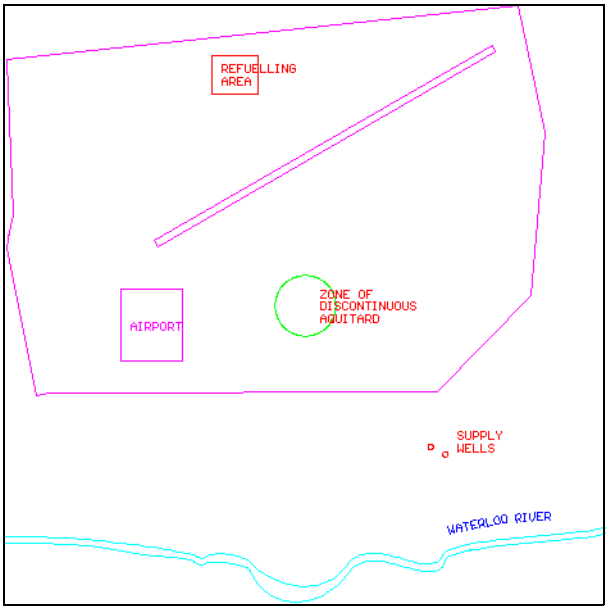
[...] - denotes a button to click on, either in a window, or in the side or bottom menu bars.

The **bold faced type** indicates menu or window items to click on or values to type in.

Some of the more significant upgrade features included in the latest versions of Visual MODFLOW are described in *italic* form.

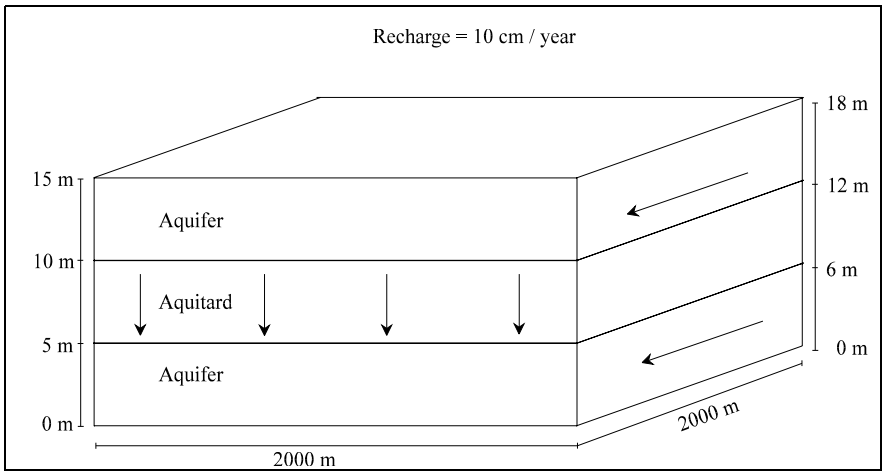
Description of the Example Model

The site is located near an airport just outside of Waterloo. The surficial geology at the site consists of an upper sand and gravel aquifer, a lower sand and gravel aquifer, and a clay and silt aquitard separating the upper and lower aquifers. The relevant site features consist of a plane refuelling area, a municipal water supply well field, and a discontinuous aquitard zone. These features are illustrated in the following figure:



The municipal well field consists of two wells. The east well pumps at a constant rate of 550 m^3/day, while the west well pumps at a constant rate of 400 m^3/day. Over the past ten years, airplane fuel has periodically been spilled in the refuelling area and natural infiltration has produced a plume of contamination in the upper aquifer. This tutorial will guide you through the steps necessary to build a groundwater flow and contaminant transport model for this site. This model will demonstrate the potential impact of the fuel contamination on the municipal water supply wells.

When discussing the site, in plan view, the top of the site will be designated as north, the bottom of the site as south, and the left side and right side as west and east, respectively. Groundwater flow is from north to south (top to bottom) in a three-layer system consisting of an upper unconfined aquifer, an intervening middle aquitard, and a lower confined aquifer, as illustrated in the following figure. The upper aquifer and lower aquifers have a hydraulic conductivity of $2e^{-4}$ m/sec, and the aquitard has a hydraulic conductivity of $1e^{-10}$ m/sec.



Module I: Creating and Defining a Flow Model

The project creation wizard was re-designed for version 4.1. Most of the default options and parameter values for flow, transport, grid, and project are now set during the initial set-up to avoid having to enter them later in various parts of the program.

Section 1: Generating a New Model

The first module will take you through the steps necessary to generate a new model data set using the Visual MODFLOW modeling environment.

On your Windows desktop, you will see an icon for Visual MODFLOW:

☞☞ to start the Visual MODFLOW program

To create your new model:

☞ **File** from the top menu bar

☞ **New**

A **Create new model** window will appear.

For your convenience, we have already created a model following this step-by-step tutorial. The model is located in the default directory **C:\My Documents\Visual MODFLOW\Tutorial**.

It is recommended that you create a new folder, and save the new model in this folder.

To create a new folder:

☞

Type in a new folder name.

☞☞ on the new folder

Type: Airport in the **File name** field

☞ **[Save]**

NOTE: If you choose the default directory (**C:\My Docuements\Visual MODFLOW\Tutorial**) and type in the same **Airport** file name, a warning message may appear if the Airport.vmf file has already been created.

- ☞ **[No]** to save your new model in a different folder, or
- ☞ **[Yes]** to overwrite the existing model.

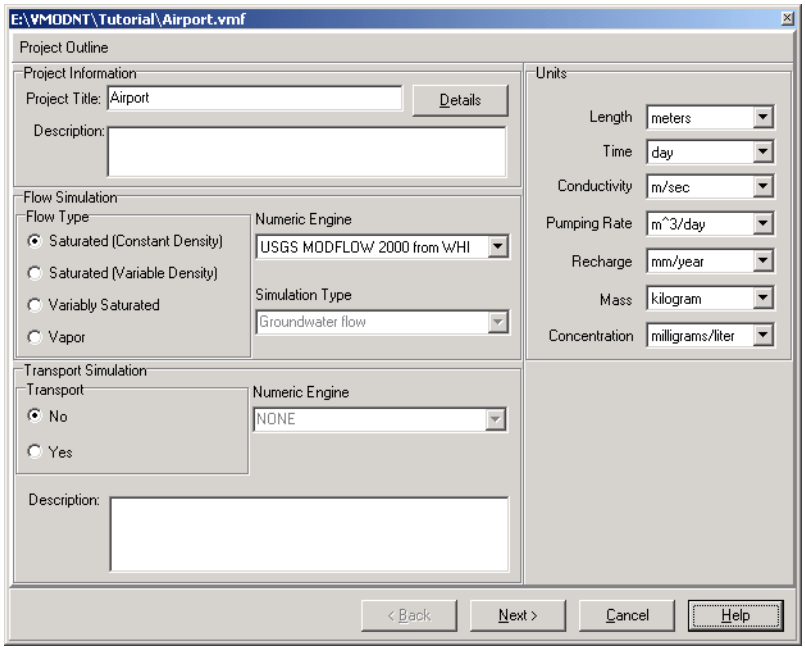
Visual MODFLOW automatically adds a .**VMF** extension onto the end of the filename.

Next, the Model Setup is described in four consecutive steps.

Step 1: Project Settings

In Step 1, the following window is used to select:

- the desired **Project Information**
- **Units** associated with various flow and transport parameters, and
- the desired **Engines** for the Flow and Transport simulation



For the Flow model, the following Numeric Engines are available:

- USGS MODFLOW-96 from WHI (saturated, constant density)
- USGS MODFLOW-2000 from WHI (saturated, constant density)
- USGS MODFLOW-2005
- USGS SEAWAT 2000 (saturated, variable density)
- MODFLOW-SURFACT from HGL (variably saturated)

In this example, **MODFLOW-2005** is selected.

Visual MODFLOW supports all of the latest versions of the public domain and proprietary programs for contaminant transport modeling, including:

- MT3DMS
- MT3Dv150
- MT3D99
- RT3Dv1.0

- RT3Dv2.5
- PHT3Dv.1.46
- PHT3Dv.2.0

Note: The MT3DMS, MT3D99 and RT3D, PHT3D transport engines allow for multi-species contaminant transport. MT3D150 allows only a single species to be modeled.

For MT3DMS transport variants that involve multiple species, Visual MODFLOW gives you the ability to leverage the power of multiple processors on your machine (or dual core processor). One or more species can be allocated to the available processes on your computer, and solved independently, effectively improving the solving performance of Visual MODFLOW.

☞ **[Yes]** radio button, to enable the Transport Simulation options

In this example, **MT3DMS** is selected by default, and will be used for the simulation.

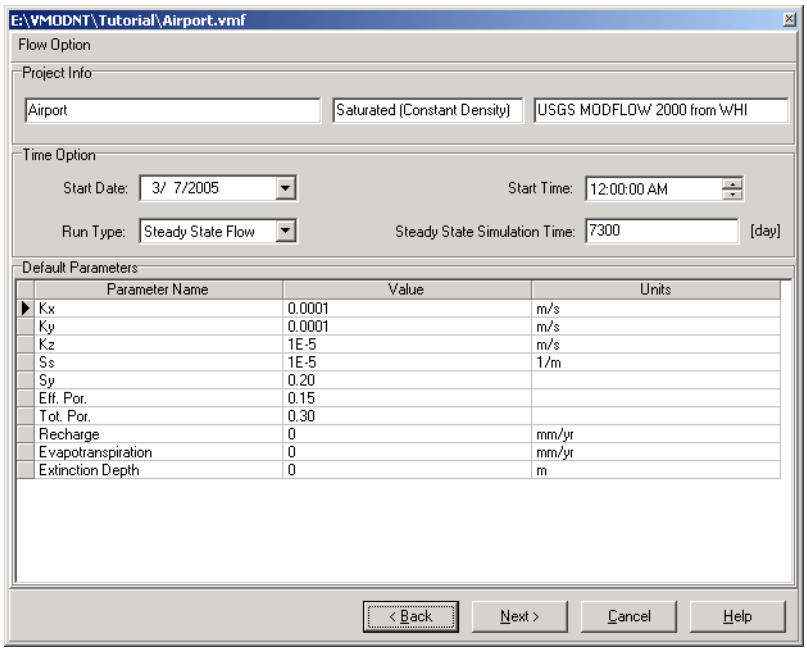
Under the **Units** section, select the following information for each model input data type.

Length:	**meters**
Time:	**day**
Conductivity:	**m/sec**
Pumping Rate:	**m^3/day**
Recharge:	**mm/year**
Mass:	**kilogram**
Concentration:	**milligrams/liter**

☞ **[Next]** to accept these values

Step 2: Flow Settings

In Step 2, the following window will appear displaying the default parameter values for the Flow simulation.



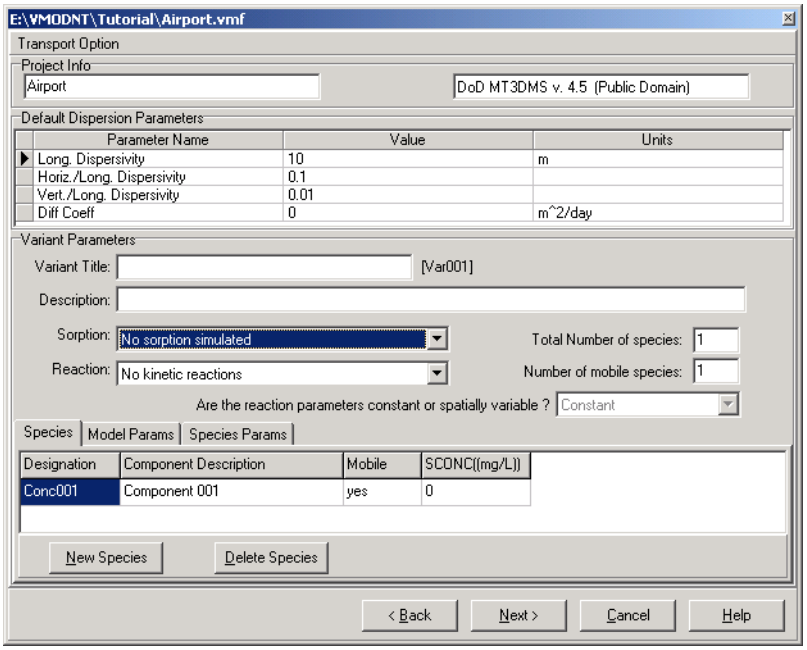
type: 7300 for the Steady State Simulation time

The **Start Date** and **Start Time** of the model are corresponding to the beginning of the simulation time period. Currently, this date is relevant only for transient flow simulations where recorded field data may be imported for defining time schedules for selected boundary conditions (Constant Head, River, General Head and Drain).

☞ **[Next]** to accept these values

Step 3: Transport Settings

In Step 3, the following window will appear displaying the default parameter values for the Transport simulation.



In order to support all of the available options for the multi-species reactive transport programs, Visual MODFLOW requires you to setup the initial conditions for the contaminant transport scenario (e.g. number of chemical species, names of each chemical species, initial concentrations, decay rates, partitioning coefficients, etc.). Each scenario is referred to as a Transport Variant, and you can have more than one variant for a given flow model.

By default Visual MODFLOW creates a variant named **VAR001**.

We will now edit this variant to setup the model for transport processes.

The default transport variant will use MT3DMS as the numeric engine, with no sorption or reaction terms.

For this tutorial you will add a linear sorption process to the transport variant.

☞ in the **Sorption** combo box

☞ Select **Linear Isotherm [equilibrium-controlled]**

For this tutorial you will not be simulating any decay or degradation of the contaminant, so the default **Reactions** settings of **No kinetic reactions** will be fine.

Below the **Sorption** and **Reaction** settings are three tabs labelled:

- **Species**

- **Model Params**
- **Species Params**

In the **Species** tab you will see a spreadsheet view with labelled column headings. The last two columns are labelled with MT3D variable names.

The chemical species being simulated in this model is JP-4. Visual MODFLOW allows you to use the real names of the simulated chemicals in order to easily identify them in contour maps and graphs.

Type the following information for the chemical species:

Type: JP-4 in the **Designation** column

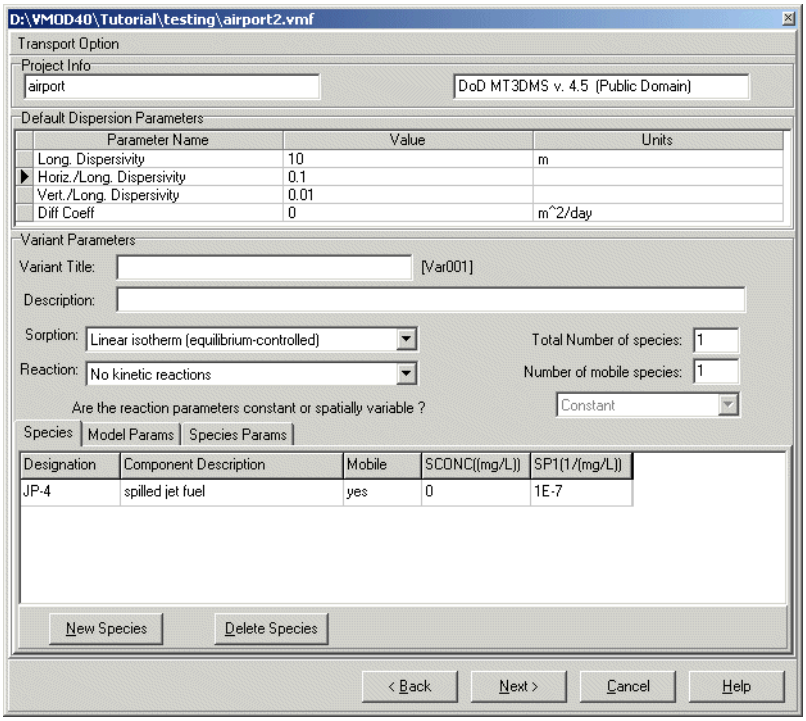
Type: Spilled Jet Fuel in the **Component Description** column

☞ in the **Mobile** column and select **Yes**

Type: 0 in the **SCONC [mg/L]** column

Type: 1e-7 in **SP1 [1/(mg/L)]** column

The species information is now set up properly, and should appear similar to the figure shown below.

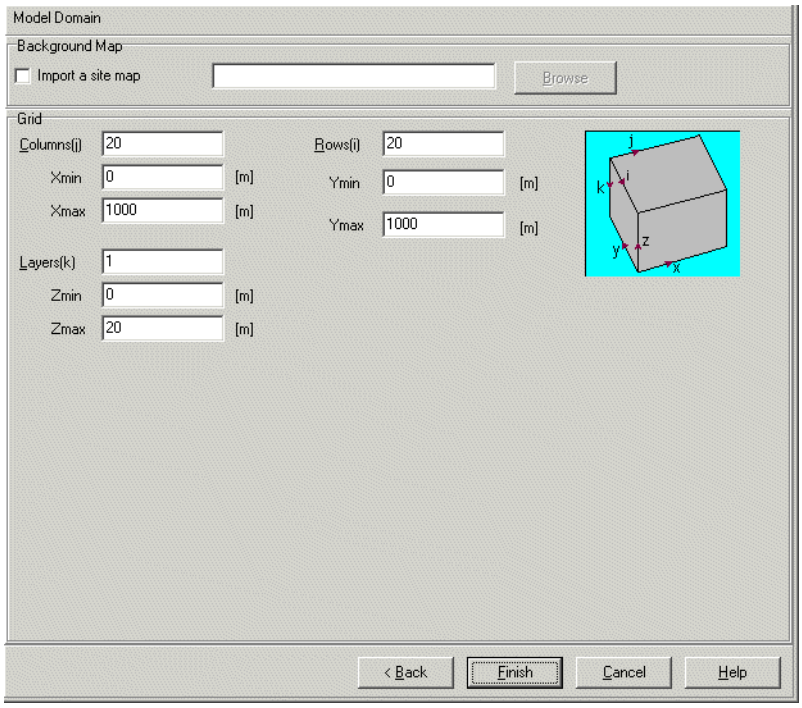


This completes the setup for the Transport Engine. You are now ready to exit the **Transport Engine Options** window.

☞ **[Next]** to continue.

Step 4: Model Domain

Step 4 is **Creating the Model Grid** (see figure below).



In this step you can **Import a site map**, specify the dimensions of the **Model Domain**, and define the number of rows, columns, and layers for the finite difference grid.

Enter the following number of rows, columns, and layers to be used in the model. **Type** the following into the **Model Domain** section,

Columns (j):	**30**
Rows (i):	**30**
Layers (k):	**3**
Zmin:	**0**
Zmax:	**18**

☞ **☑ Import a site map**

Next, you must specify the location and the file name of the **.DXF** background map.

☞ **[Browse]**

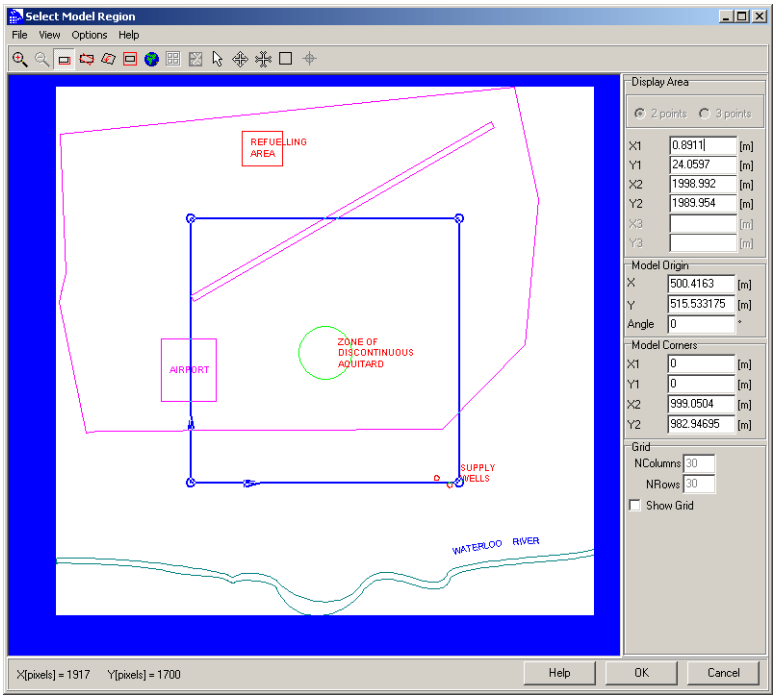
Navigate to the **My Documents\Visual MODFLOW\Tutorial\Airport** directory and select the following file:

☞ **Sitemap.dxf**

☞ **[Open]**

☞ **[Finish]** to accept these settings

A **Select Model Region** window will appear prompting you to define the extents of the model area. Visual MODFLOW will read the minimum and maximum co-ordinates from the site map (**Sitemap.dxf**) and suggest a default location that is centred in the model domain.



Type the following values over the numbers shown on the screen:

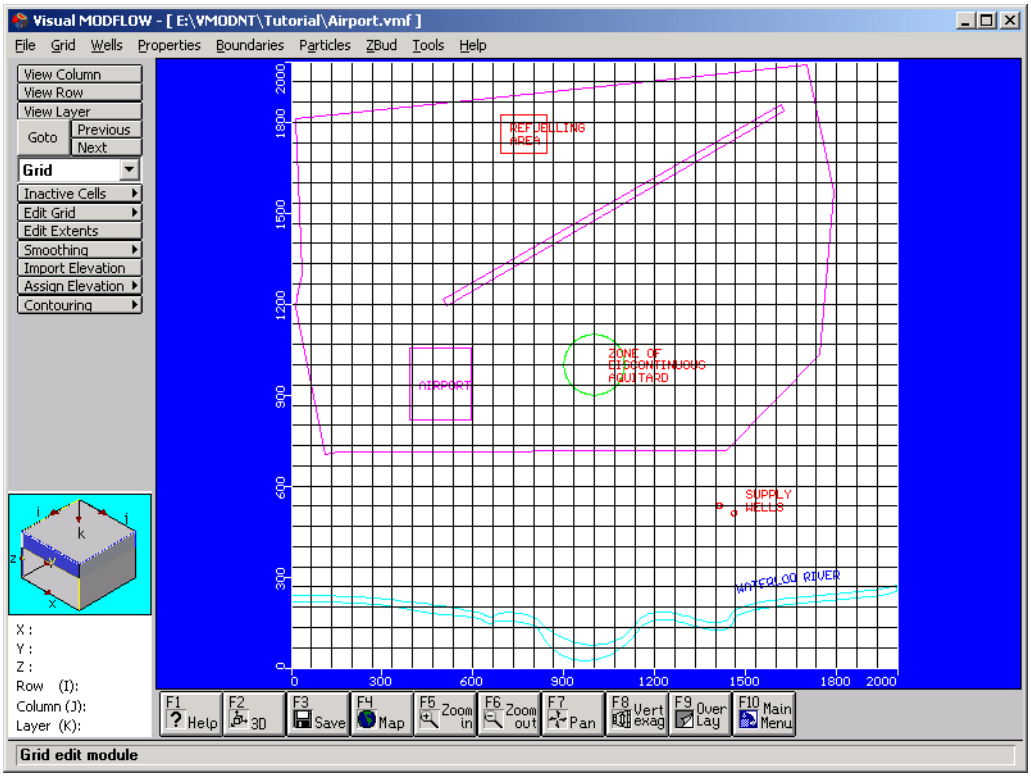
Display area:	**X1:**	**0** ⇔
	Y1:	**0** ⇔
	X2:	**2000** ⇔
	Y2:	**2000** ⇔
Model Origin:	**X:**	**0** ⇔
	Y:	**0** ⇔
	Angle:	**0** ⇔
Model Corners:	**X1:**	**0** ⇔
	Y1:	**0** ⇔
	X2:	**2000** ⇔
	Y2:	**2000** ↵

☞ **[OK]** to accept the model dimensions.

A **File attributes** window will appear to indicate that the "Sitemap.dxf" is being saved inside the Visual MODFLOW project with the name "Airport.Sitemap.MAP".

☞ **[OK]**

The **Input** menu will open, and a uniformly spaced 30 x 30 x 3 finite-difference grid will automatically be generated within the model domain. The Grid and site map will appear on the screen, as shown in the following figure:



By default the **Grid** screen is loaded when you first enter the **Input** module.

Add Air Photo

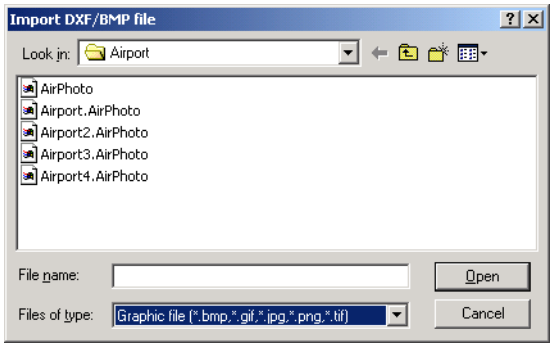
At this time, you may add an additional map for improved visualization; you will add an airphoto of the site, in bitmap format.

- ☞ **F4 (Map)** button from the bottom toolbar.

A dialog will appear where you can specify the file to import. Browse to **My Documents\Visual MODFLOW\Tutorial\Airport**.

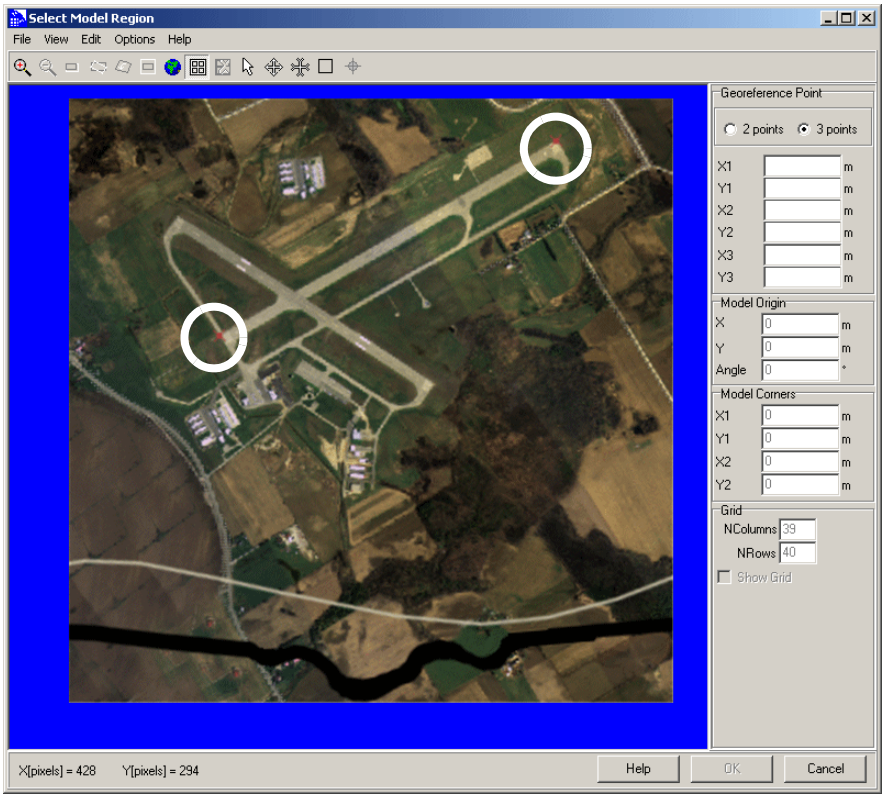
At the bottom of the dialog, under Files of Type,

- ☞ **Graphics File** from the list.
- ☞ **Airport.Airphoto.bmp** file from the list.



- ☞ **[Open]**

A “Select Model Region” dialog will then appear, as shown below.

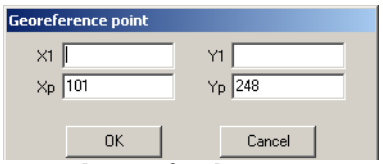


In order to map the pixels of the image to a coordinate system, the image must have two georeference points with known coordinates. These georeference points can be defined using the procedure described below. For this image, the two points have been marked with a red X, and are located at opposite ends of the airport’s main runway (and are circled in the image above).

To help compensate for image stretching and skewing, Visual MODFLOW allows you to georeference images using three georeference points.

To set the georeference points,

- Click on the red X located on the **left** side of the map. A **Georeference point** dialog will appear prompting for the X1 and Y1 world coordinates of the selected location:



- Enter the **X1** and **Y1** coordinates for this point.

  for **X1**, type: 502

  for **Y1**, type: 1212

- Click **[OK]**

- Click on the red X located on the **right** side of the map. A **Georeference point** dialog will appear prompting for the X2 and Y2 world coordinates of the selected location:

  for **X2**, type: 1619

  for **Y2**, type: 1863

- Click **[OK]**

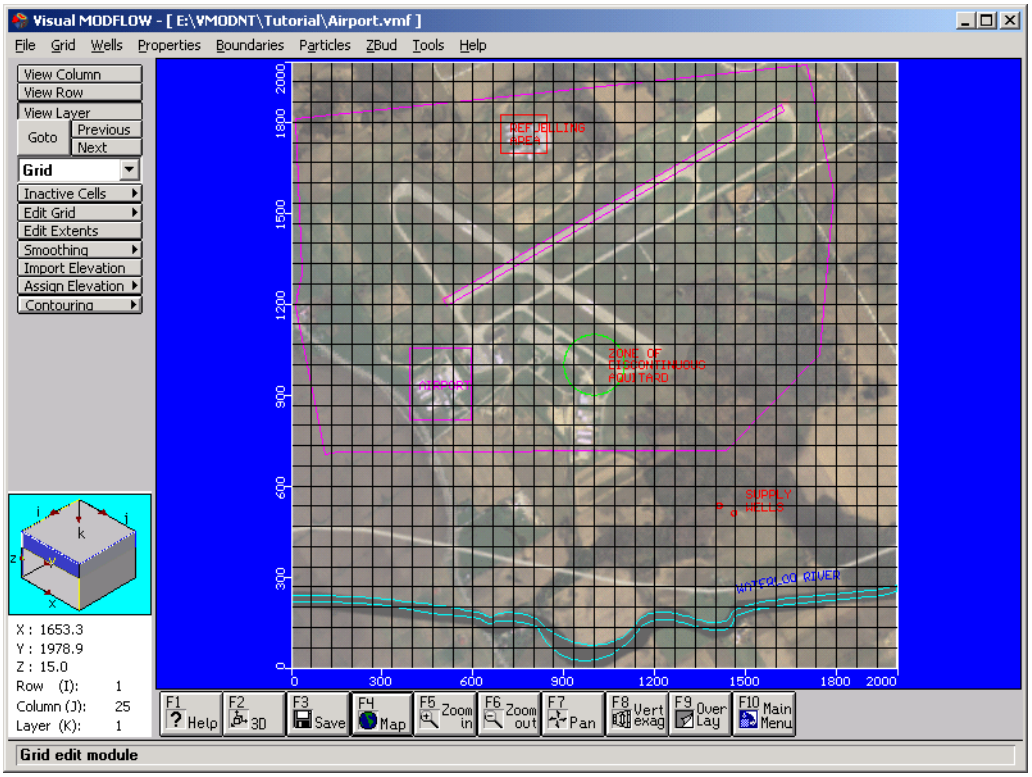
A box will appear around the map region.

- Click **[OK]** to continue

A confirmation message will appear, displaying the filename of the georeferenced image.

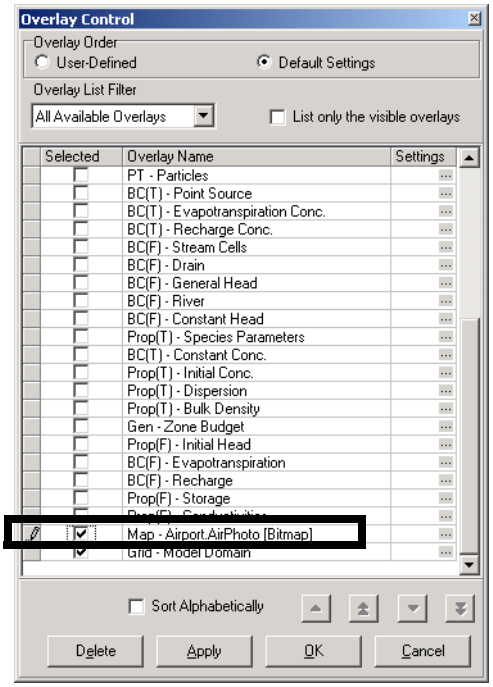
- Click **[OK]** to continue

If the map image was correctly georeferenced, the river and runway locations should match up closely with those digitized in the AutoCAD Sitemap.dxf file. The display should now be similar to the one shown below.



The site map properties can be defined in the overlay control.

☞ **F9 (Overlay Control)** button from the bottom toolbar.



For the remainder of this tutorial, the screenshots will not display the airphoto; you may keep this visible if you wish.

- ☞ **Map-Airport.Airphoto [Bitmap]** checkbox, to turn off this map.
- ☞ **[OK]**

This concludes Section 1: Generating a new Model. In the next section, you will refine the model grid.

Section 2: Refining the Model Grid

The **Grid** screen provides a complete assortment of graphical tools for refining the model grid, delineating inactive grid zones, importing layer surface elevations, assigning elevations, optimizing (smoothing) the grid spacing, and contouring the layer surface elevations.

This section describes the steps necessary to refine the model grid in areas of interest, such as around the water supply wells, refueling area, and area of discontinuous aquitard. The reason for refining the grid is to get more detailed simulation results in areas of interest, or in zones where you anticipate steep hydraulic gradients. For example, if drawdown is occurring around the well, the water table will have a smoother surface if you use a finer grid spacing. Also, layer properties can be assigned more correctly on a finer grid.

To refine the grid in the X-direction,

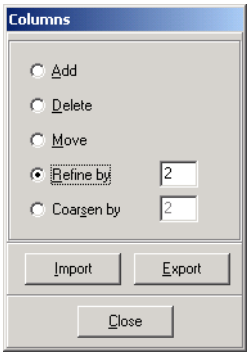
- ☞ **[Edit Grid >] Edit Columns**

A **Columns** window will appear showing the options you have for editing the grid columns. By default the **Add** option is selected, and new grid lines can be added by clicking the mouse on the desired location of a new grid line.

Move the mouse anywhere in the grid. Notice that a highlighted vertical line follows the location of the mouse throughout the model grid. This line may be used to add a column at

any location within the model domain. In this exercise, you will refine the model grid in the refueling area, near the water supply wells, and around the discontinuous aquitard border.

☞ ⊙ **Refine by**



Leave the default value of 2 as the refining criterion.

Click anywhere on the blue background to activate the model domain. Move the mouse cursor over the model and click at following X-coordinates (do not worry if you cannot hit the exact values, simply click on the column closest to this number):

Hint: Use navigation cube for assistance.

Region 1:

- X = 670
- X = 940

Region 2:

- X = 1070
- X = 1140

Region 3:

- X = 1340
- X = 1600

As you click on any pair of columns, each interval between them is divided into two by an extra column.

☞ **[Close]** to exit the **Columns** window

Next, you will refine the grid in the Y-direction around the refuelling area, the area of discontinuous aquitard, and the supply wells.

☞ **[Edit Grid >] Edit Rows**

☞ ⊙ **Refine by**

Leave the default value of 2 as the refining criterion.

Click anywhere on the blue background to activate the model domain. Move the mouse cursor over the model and click at following Y-coordinates (do not worry if you cannot hit the exact values, simply click on the column closest to this number):

Region 1:

- Y = 400
- Y = 730

Region 2:

- Y = 860
- Y = 930

Region 3:

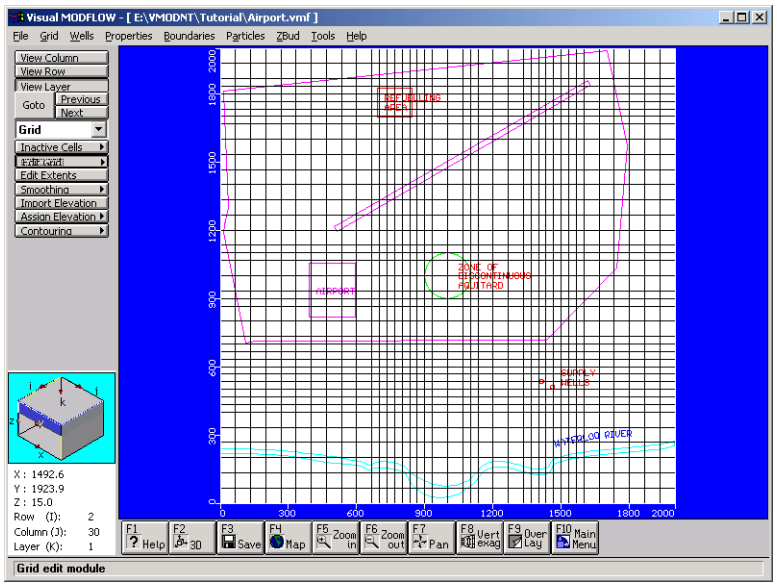
- Y = 1060
- Y = 1130

Region 4:

- Y = 1660
- Y = 1860

☞ **[Close]** to exit the **Rows** window

The refined grid should appear as shown in the following figure:



The following steps will show you how to view the model in cross-section:

☞ **[View Column]** from the left toolbar

Move the mouse cursor anywhere in the grid. As you move the cursor across the screen, a red bar highlights the column corresponding to the cursor location. To select a column to view,

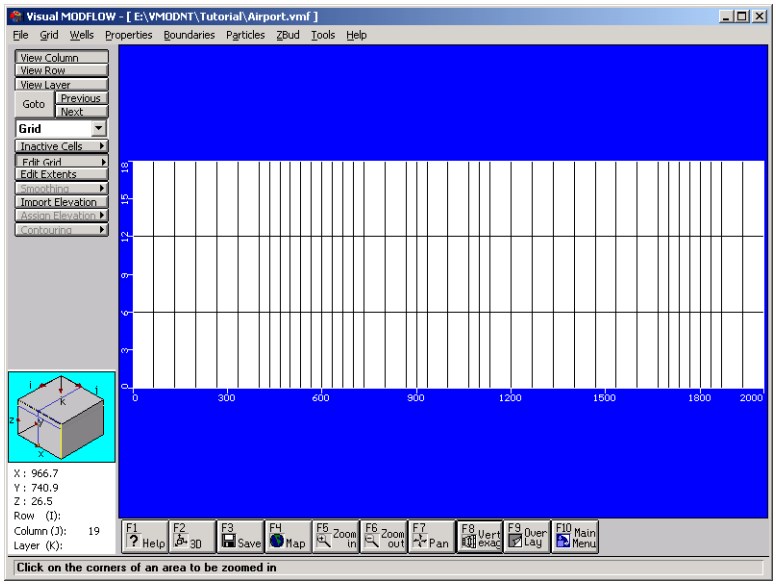
☞ Left mouse button on the desired column

Visual MODFLOW transfers the screen display from plan view to a cross-sectional view of the model grid. At this point the model has no vertical exaggeration and the cross-section will appear as a thick line with the three layers barely discernible. To properly display the three layers, you will need to vertically exaggerate the cross-section.

☞ **[F8 - Vert Exag]** from the bottom of the screen. A **Vertical Exaggeration** window appears prompting you for a vertical exaggeration value.

Type: 40

☞ **[OK]** The three layers of the model will then be displayed, as shown below:



From the figure you can see that each layer has a uniform thickness across the entire cross-section. However, nature rarely provides us with geology that conforms to flat horizontal layering with uniform layer thickness.

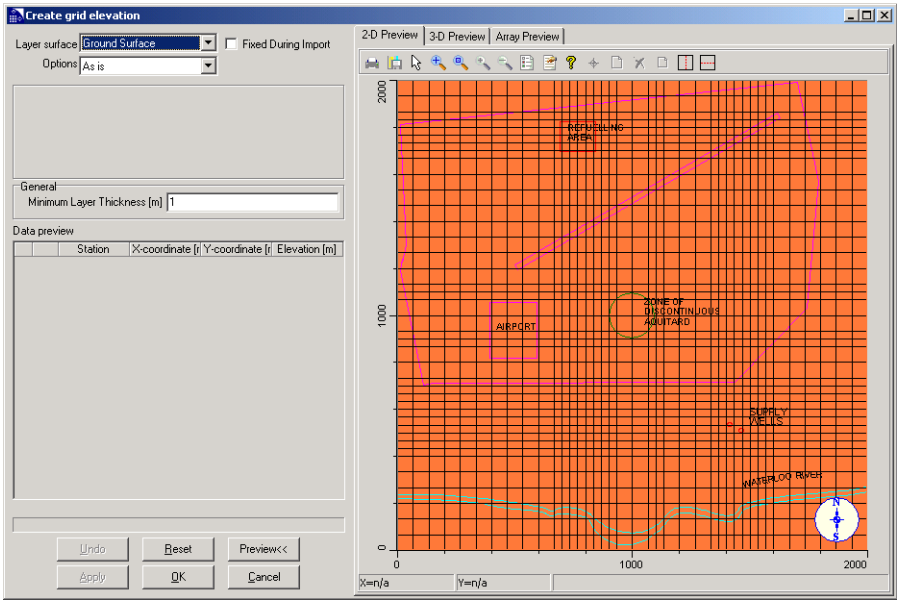
Visual MODFLOW provides a dramatically improved set of tools for importing, creating, and modifying layer surface elevations for the finite difference model grid. From importing USGS DEM files, to specifying a constant slope using Strike and Dip information, no other graphical interface offers a more complete set of features for designing the model grid layer surfaces. Visual MODFLOW supports importing of variable layer elevations from a variety of data files including

- Points data (XYZ) ASCII files (.TXT, .ASC, .DAT),
- MS Access Database files (.MDB),
- MS Excel files (.XLS),
- ESRI Point files (.SHP),
- USGS DEM files (.DEM),
- Surfer grid files (.GRD),
- ESRI grid files (.GRD), and
- Mapinfo grid files (.GRD).

In this example, we will import text files containing elevation data for selected X and Y coordinates within the model domain.

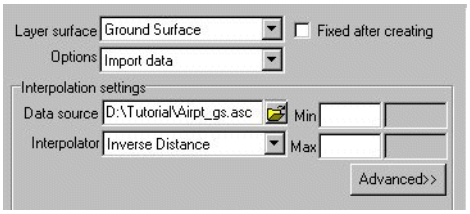
☞ **[Import Elevation]** from the left-hand toolbar

The following **Create grid elevation** wizard will appear:



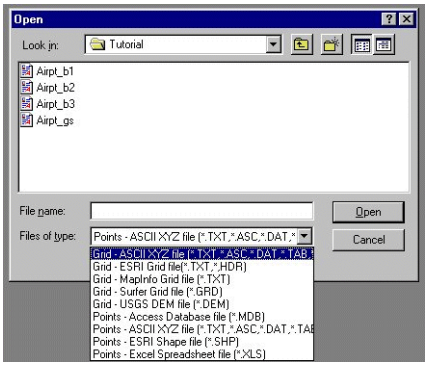
This wizard allows you to import and interpolate a data set to the model grid for each of the selected model layers. By default, in the **Layer surface** field, the **Ground Surface** is the selected layer. **Previews** of the **2-D** and **3-D** interpolated/modified layer surface elevations, and a cell-by-cell spreadsheet data **Array**, are displayed in tabs on the right side of the window.

☞ in the **Options** field and select **Import data** from the pull-down menu.



Upon selecting this option, the **Interpolation settings** frame will appear underneath the **Options** field. It is used to select a data file, and an interpolation method (Natural Neighbors, Kriging, or Inverse Distance). By default, in the **Interpolator** field, **Natural Neighbors** is selected. However, for this example, use the button to select **Inverse Distance**.

☞ associated with the **Data source** field, and the following **Open** window will appear:

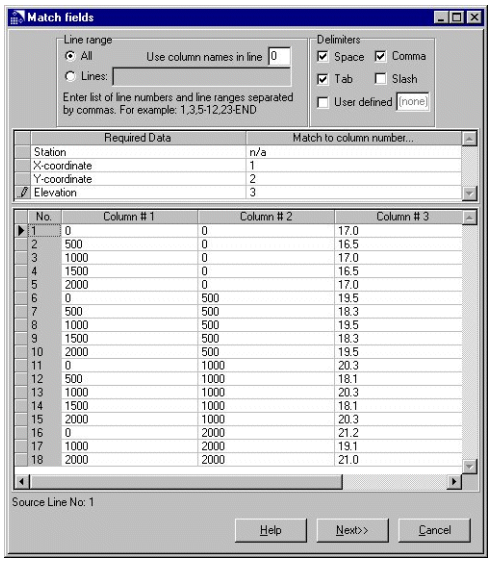


To select the ground surface topography data file, browse to **My Documents\Visual MODFLOW\Tutorial\Airport** and,

☞ **Airpt_gs.asc** (accept the default file format)

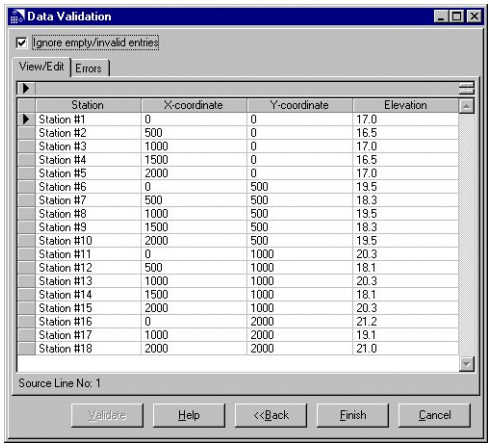
☞ **[Open]**

The following **Match fields** window will appear.



The **Required Data** frame lists the columns from the data file, and the **Match to column number** frame is used to match the corresponding data to each data field, as required by the wizard. As shown in the above window, **Type** the appropriate number in each field to match **Column #1** to **X-coordinate**, **Column #2** to **Y-coordinate**, and **Column #3** to **Elevation**. Once all columns are matched, the **[Next]** button is activated.

☞ **[Next]** to proceed to the **Data Validation** window (shown below).

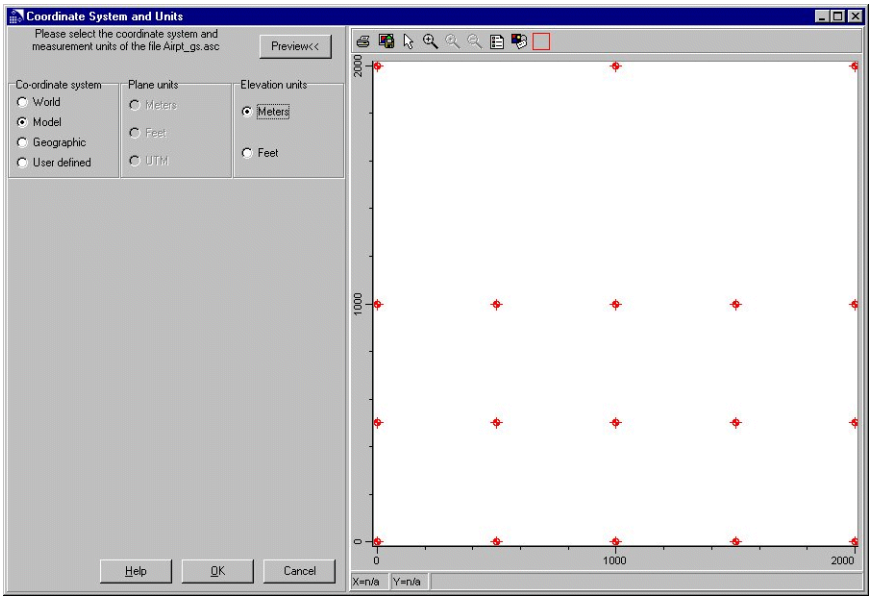


Since no invalid entries (offending cells would be highlighted with a red color) are detected, it ensures that the data set contains valid data, and that data points are located within the model domain, as depicted in the **Coordinate System and Units** window shown below.

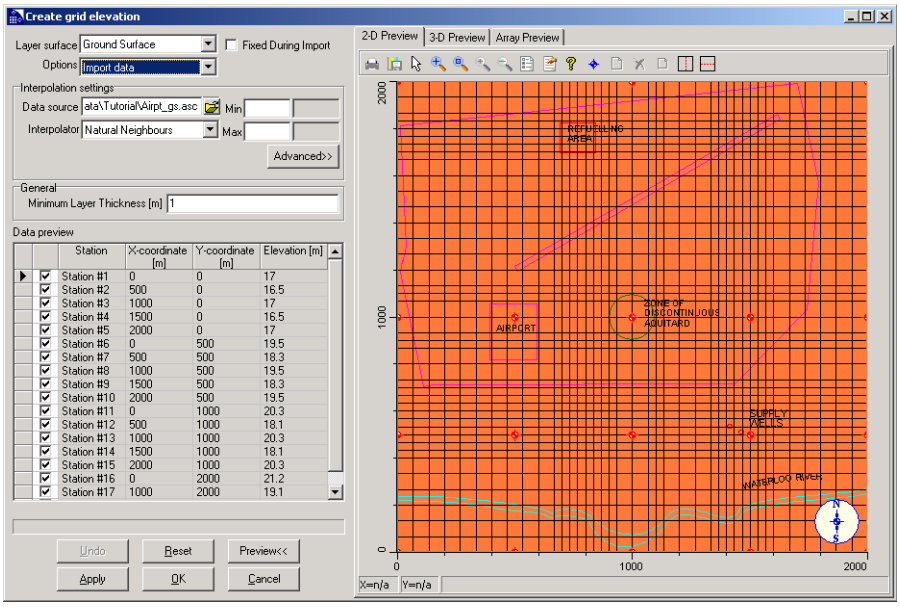
☞ **[Finish]** to proceed to the **Coordinate System and Units** window, and then

☞ Select **Model** in the **Coordinate System** frame, and **Meters** in the **Elevation Units** frame (as shown below).

☞ Click **[Preview]** button

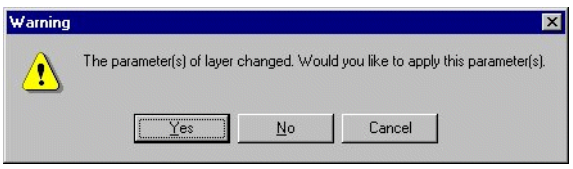


☞ **[OK]** and the following window will appear.



☞ **[OK]**

Before proceeding further and assigning the interpolated layer elevations to the selected model grid layer, a **Warning** message appears to confirm that the model elevations are going to be changed.



☞ **[Yes]**

If the proposed changes to your model surface elevations would conflict with the head values of previously assigned boundary conditions, a warning message would appear, providing you with the opportunity to address the issue. In this case, no potential problems are detected.

Next you will import the surface elevations for the bottom of Layer 1.

- **[Import Elevation]**
- associated with the **Layer surface** field and select **Bottom of Layer 1** from the pull-down menu.
- associated with the **Options** field and select **Import data** from the pull-down menu.
- to select the **Inverse Distance** interpolation method
- associated with the **Data source** field and select the **Airpt_b1.asc** from the **My Documents\Visual MODFLOW\Tutorial\Airport** folder (to select the elevation data for the bottom of Layer 1) in the **Open** window.
- **[Open]**

Match **Column #1** to the **X-coordinate**, **Column #2** to the **Y-coordinate**, and **Column #3** to the **Elevation**.

- **[Next]** to view the **Data Validation** window.
- **[Finish]** to proceed to the **Coordinate System and Units** window.
- Select **Model** in the **Coordinate System** frame, and **Meters** in the **Elevation Units** frame.
- **[OK]** to import the elevation data.
- **[Apply]** in the **Create grid elevation** window.

Next you will import the surface elevations for the bottom of Layer 2 and Layer 3.

- associated with the **Layer surface** field and select **Bottom of Layer 2** from the pull-down menu.
- associated with the **Options** field and select **Import data** from the pull-down menu.
- to select the **Inverse Distance** interpolation method
- associated with the **Data source** field and select the **Airpt_b2.asc** from the **My Documents\Visual MODFLOW\Tutorial\Airport** folder (to select the elevation data for the bottom of Layer 2) in the **Open** window
- **[Open]**

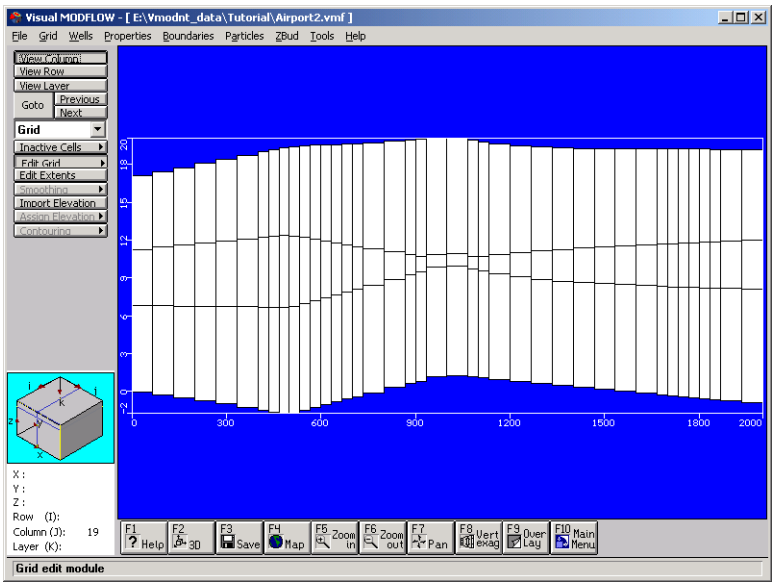
Match **Column #1** to the **X-coordinate**, **Column #2** to the **Y-coordinate**, and **Column #3** to the **Elevation**.

- **[Next]** to view the **Data Validation** window.
- **[Finish]** to proceed to the **Coordinate System and Units** window, and then select **Model** in the **Coordinate System** frame, and **Meters** in the **Elevation Units** frame.
- **[OK]** to import the elevation data.
- **[Apply]** in the **Create grid elevation** window.
- **[Yes]** to confirm the changes in the model elevations.

The bottom of layer 2 will then appear with a variable layer surface.

Next you will import the surface elevations for the bottom of Layer 3. Follow the previously described process, noting that for the **Bottom of Layer 3**, you will select **Airpt_b3.asc** from the pull-down menu. The bottom of Layer 3 will then appear with a variable layer surface.

The model cross-section should appear similar to the following figure (when viewing column 19).



Next you will return to the layer view of the model domain,

- ☞ **[View Layer]** from the left toolbar

Then highlight the top layer of the model and click left mouse button. This will create a plan view of the Airport site.

Section 3: Adding Pumping Wells

The purpose of this section is to guide you through the steps necessary to assign pumping wells to the model.

- ☞ **[Wells]** from the top menu bar
- ☞ **[Pumping Wells]** from the drop-down menu

You will be asked to save your data:

- ☞ **[Yes]** to continue

Once your model has been saved, you will be transferred to the **Pump Wells** input screen. Notice that the left-hand toolbar buttons now contain well-specific options for importing, adding, deleting, editing, moving, and copying pumping wells.

Before you begin adding the pumping wells for this model, zoom in on the area surrounding the water “Supply Wells” text (in the lower right-hand section of the model domain).

- ☞ **[F5 - Zoom In]**

Move the cursor to the upper right of the “Supply Wells” text and click on the left mouse button. Then stretch a zoom window around the supply wells location markers, and click again to view the selected area.

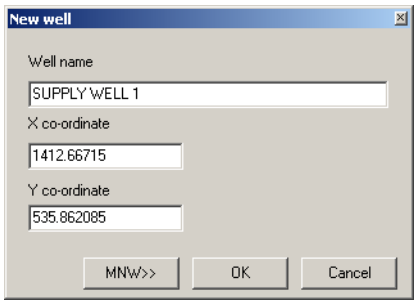
To add a pumping well to your model,

☞ **[Add Well]**

Move the pointer to the western pumping well location marker and click the left mouse button to add a pumping well at this location.

The Visual MODFLOW well manager has been dramatically improved to allow for easier definition and modification of properties for pumping wells and observation wells. The new interface allows you to view all pumping wells / observation wells in a single table, screen wells between selected model layers and easily find wells using the Find utility. For more information, please refer to the Visual MODFLOW User’s Manual.

A **New Well** dialog will appear (shown below) prompting you for a well name.



Type the following information:

Well Name: Supply Well 1 ⇔

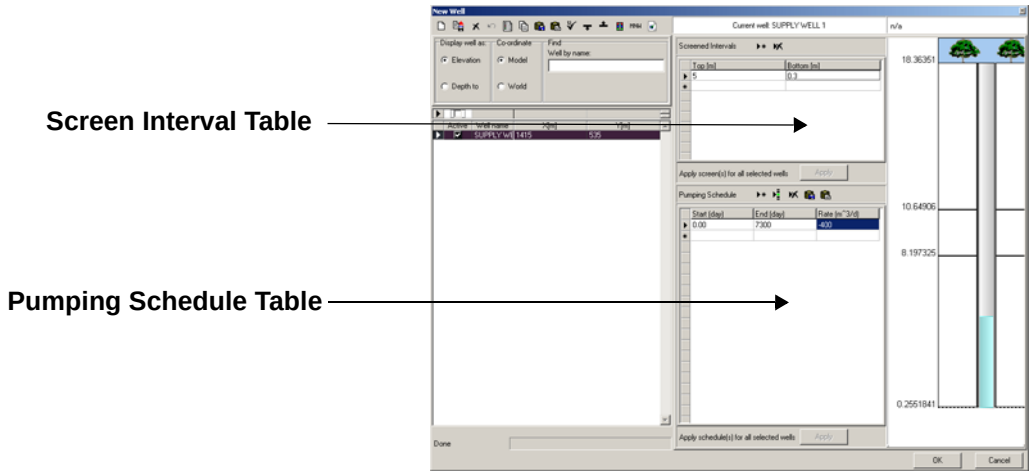
Overwrite the existing coordinate information by typing the following:

X: 1415 ⇔

Y: 535

☞ **[OK]**

The following window will display, prompting you to enter Screen Interval and Pumping Schedule data for the new well:



To add a **Screened Interval**,

In the Screen Interval Table, click in the column labelled **Bottom** and **Type** the following values:

Screen Top (m): **5.0** ⇔

Screen Bottom (m): **0.3**

Notice the well screen interval appears in the well bore diagram on the right-hand side of the window. The well screen interval can be modified by clicking-and-dragging the edge of the well screen to a new top or bottom elevation.

To enter the well **Pumping Schedule**, click the left mouse button once inside the text box under the column labelled **End (day)** and **Type** the following information.

End (day): **7300** ⇔

Rate (m^3/day): **- 400**

☞ **[OK]** to accept this well information.

An end time of 7300 days was used because this simulation will be run for twenty years. However, since this will be a steady-state flow simulation, you can use any time value you like and the results for the flow simulation will not change. A Steady State run means that the model is run using the first input value for each parameter until equilibrium is achieved, irrespective of relative time. The rate is entered as a negative number because this well is an extraction well.

If you have failed to enter any required data, Visual MODFLOW will prompt you to complete the table at this time.

The next step is to assign the well parameters for the second pumping well by using a shortcut. This will allow you to copy the characteristics from one well to another location.

☞ **[Copy Well]** from the left toolbar

Move the pointer on top of the western well (Supply Well 1) and click on it. Then move the cursor to the location of the eastern well and click again to copy the well.

Next you will edit the well information from the new (copied) well.

☞ **[Edit Well]** from the left toolbar and select the new pumping well, currently named "Supply Well 1 (2)"

An **Edit Well** window will appear. In this window:

Type: Supply Well 2 in the **Well Name** field

Type: X=1463, Y= 509, and

Type: -550 in the **Rate (m^3/day)** field

☞ **[OK]** to accept these changes

☞ **[F6 - Zoom Out]**

The USGS drawdown-limited Multi-Node Well (MNW) package has been integrated with Visual MODFLOW to allow for simulation of intraborehole flow in wells with screens that span multiple aquifer layers. Through the newly improved Wells Manager window, you can choose which well package to use (WEL or MNW), specify the well loss type (Skin, Linear or Nonlinear) and its respective values to simulate well losses, specify drawdown-pumping constraints and select the cell-to-well conductance distribution method. For more information on this new feature, please refer to the Visual MODFLOW Users' Manual.

Section 4: Assigning Flow Properties

This section will guide you through the steps necessary to design a model with layers of highly contrasting hydraulic conductivities.

☞ **Properties/Conductivity** on the top menu bar

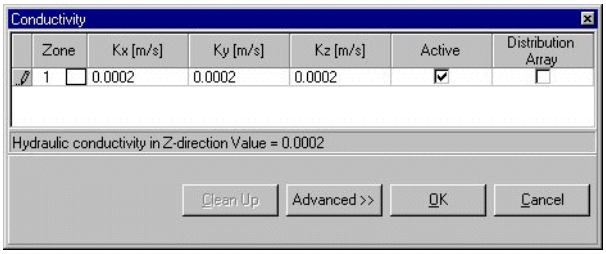
☞ **[Yes]** to save your data

You will now be transferred to the **Conductivity** input screen where you will modify the default hydraulic conductivity values (Kx, Ky, and Kz) for each cell in the model.

Click on the **[Database]** button and **Type** the following values in the window:

Kx (m/s): **2e-4** ⇔
Ky (m/s): **2e-4** ⇔
Kz (m/s): **2e-4**

☞ **[OK]** to accept these values



Note that in this case the Kx, Ky, and Kz values are the same, indicating the assigned property values are assumed horizontally and vertically isotropic. However, anisotropic property values can be assigned to a model by modifying the Conductivity Database.

In this three layer model, layer 1 represents the upper aquifer, and layer 3 represents the lower aquifer. Layer 2 represents the aquitard separating the upper and lower aquifers. For this example, we will use the previously assigned hydraulic conductivity values (**Zone# 1**) for model layers 1 and 3 (representing the aquifers) and assign different Conductivity values (i.e. a new Zone) for model layer 2 (representing the aquitard). Note that layer 1 is the top model layer.

☞ **[Goto]** from the left toolbar

The **Go To Layer** window will appear.

Type: 2

☞ **[OK]**

You are now viewing the second model layer, representing the aquitard. The next step in this tutorial is to assign a lower hydraulic conductivity value to the aquitard (layer 2). We can graphically assign the property values to the model grid cells.

The **[Assign >] Window** button in the left-hand toolbar allows you to assign a different hydraulic conductivity inside a rectangular window.

☞ **[Assign]**

☞ **[Window]**

Move the mouse to the north-west corner of the grid and click on the centre of the cell. Then move the mouse to the south-east corner and click on the centre of the cell. This creates a (white) window covering the entire layer.

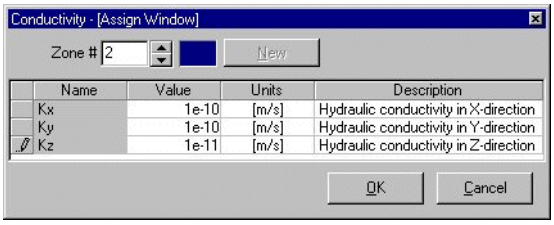
A **Conductivity - [Assign Window]** dialog will appear.

☞ **[New]**

The entire grid will turn blue and the **Zone #** will increment to a value of **2**.

To enter aquitard conductivity values (for the new **Zone# 2**), click in the **Kx Value** field and **Type** the following values:

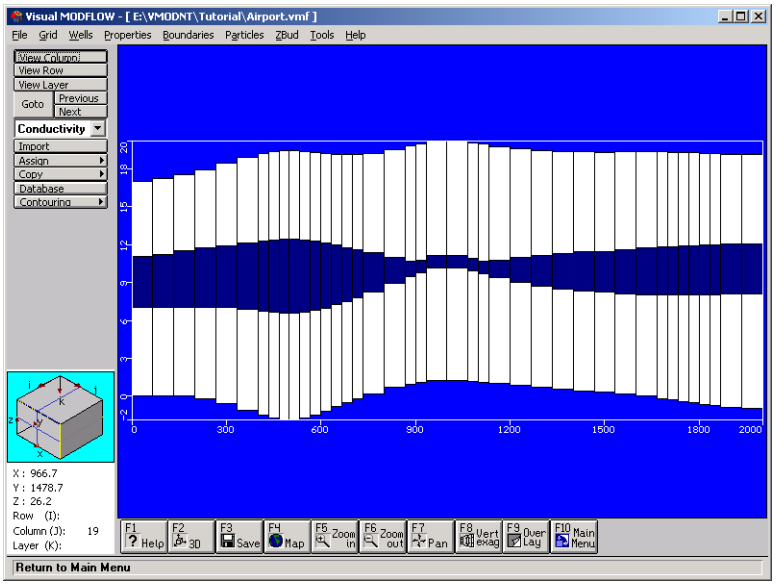
Kx(m/s): 1e-10 ⇔
Ky(m/s): 1e-10 ⇔
Kz(m/s): 1e-11



☞ **[OK]** to accept these values

Now view the model in cross-section to see the three hydrogeological units.

☞ **[View Column]** and click on a Column through the *Zone of Discontinuous Aquitard*.



The upper aquifer, aquitard, and lower aquifer units should be plainly visible by the different colors of the Conductivity property zones, as shown in the previous figure (viewing column 19).

Although the region where the aquitard pinches out is very thin, the conductivity values of these grid cells should be set equal to the Conductivity values of either the upper or lower aquifers.

To do this, you must return to the planar view of the model.

☞ **[View Layer]** and click on Layer 2

In this particular example, the zone of discontinuous aquitard is indicated on the DXF sitemap.

Next, **Zoom in** on the area where the green circle delineates the discontinuity in the aquitard.

☞ **[F5 - Zoom In]**

Move the mouse to the upper left of the discontinuous aquitard zone and click the left mouse button. Then stretch a box over the area and click again. A zoomed image of the discontinuous aquitard zone should appear on the screen.

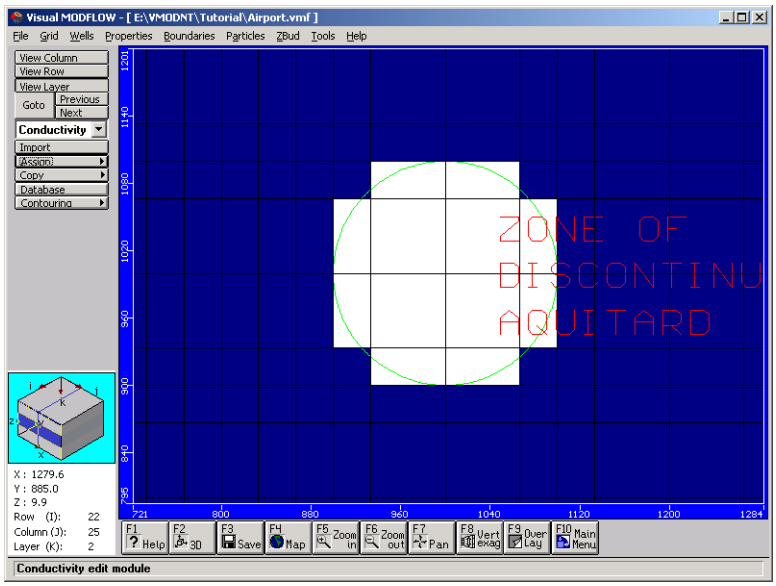
☞ **[Assign >]**

☞ **[Single]** to assign a property to an individual cell

A **Conductivity - [Assign Single]** window will appear displaying the last K property that was entered (**Zone # 2**). Click on the up-arrow key beside the Zone # field so that **Zone # 1** is the active K property.

DO NOT SELECT **[OK]** AT THIS TIME.

Move the mouse into the discontinuous aquitard area defined by the green circle. Press the mouse button down and drag it inside the area until the cells within the zone area are shaded white (as shown in the following figure).



If you accidentally re-assign some grid cells that shouldn't be included, **Right-Click** on these cells to change them back to the original setting.

Once you are finished painting the cells, click the **[OK]** button in the **Conductivity - [Assign Single]** window.

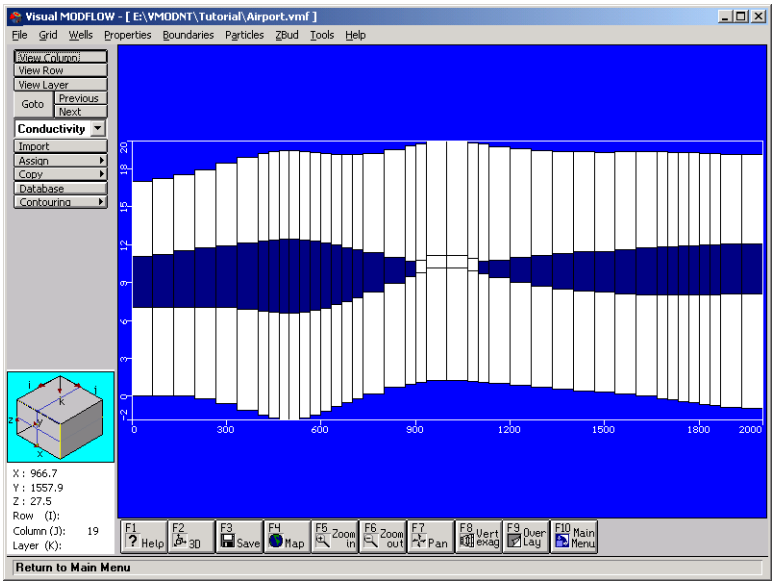
☞ **[F6 - Zoom Out]** to return to the full-screen display of the entire model domain.

To see the model in cross-section,

☞ **[View Column]**

☞ **Click** on a column passing through the *zone of discontinuous aquitard* (e.g. col. 19)

A cross-section will be displayed as shown in the following figure.



Return to the plan view by selecting the **[View Layer]** button, and then clicking on Layer 1 in the model cross-section view.

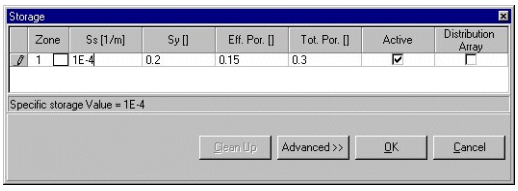
Next you will modify the storage values.

☞ **Properties/Storage** on the top menu bar

You will be transferred to the **Storage** input screen where you will modify only the default Specific Storage value, but NOT the Specific Yield, Effective Porosity, or Total Porosity values.

☞ **Database** from the left-hand toolbar

Type: 1E-4 for Ss(1/m) (as shown in the figure below):



☞ **[OK]** to accept these values

This concludes the section of the tutorial explaining how to assign constant value property zones. For this example you have modified the Conductivity and Storage values for the model. Visual MODFLOW also provides the same types of graphical tools for editing the

Initial Heads estimates for the flow model, and the model properties associated with flow and transport processes.

Section 5: Assigning Flow Boundary Conditions

The following section of the tutorial describes the steps required to assign various model boundary conditions.

Visual MODFLOW now supports defining flow boundary conditions by importing standard MODFLOW packages. The following packages can currently be imported into an existing Visual MODFLOW project: HFB1, HFB6, DRN, CHD, RIV, GHB, LAK, and WEL.

Visual MODFLOW now supports the LAK package designed to simulate lake-aquifer interactions and the ETS1 package which simulates evapotranspiration using a user-defined multi-segmented function relating evapotranspiration rate to hydraulic head.

Assigning Aquifer Recharge to the Model

In most situations, aquifers are recharged by infiltrating surface water. In order to assign recharge in Visual MODFLOW, you must be viewing the top layer of the model. Check the Navigator Cube in the lower left-hand side of the screen to see which layer you are currently in. The first boundary condition to assign is the recharge flux to the aquifer.

☞ **Boundaries/Recharge** from the top menu bar

If you are working on a new model, a **Recharge - [Assign Default Boundary]** window will prompt you to enter the **Stop Time**. Note that when the model was created, a zero recharge rate was assigned to the model as default value. Visual MODFLOW automatically assigns this recharge value to the entire top layer of the model. Enter the following information in the **Recharge** window**:**

Stop Time [day]: **7300** ⇔

Recharge [mm/yr]: **100**

☞ **[OK]** to accept these values

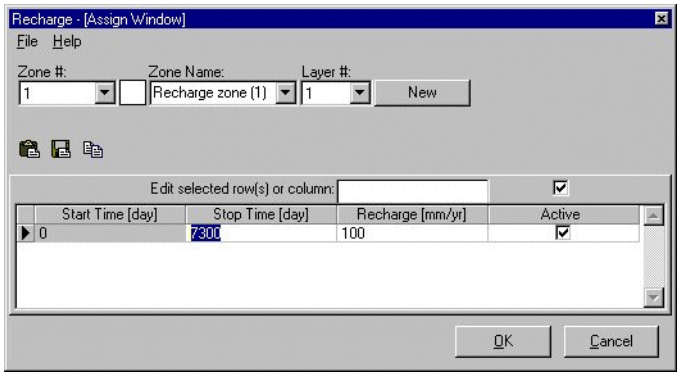
☞ **[F5-Zoom-In]**

Move the cursor to the upper left of the refuelling area and click on the left mouse button. Then stretch a box around the refuelling area and click again to close the zoom window.

Now you will assign a higher recharge value at the Refuelling Area where jet fuel has been spilled on a daily basis:

☞ **[Assign >] Window**

Move the cursor to the top-left corner of the Refuelling Area and click the left mouse button, drag a window across the refuelling area, and click the left mouse button again. The **Recharge- [Assign Window]** window will appear.



☞ **[New]** to create a new Recharge zone

The **Zone #** will increment to a value of **2** and the zone color preview box will turn blue. **Type** the following values:

Stop Time [day]: **7300** ⇔

Recharge [mm/y]: **250**

☞ **[OK]** to accept these values

The grid cells in the Refuelling Area will be colored blue to indicate they contain values for Recharge **Zone #2**.

Now return to the full view of Layer 1.

☞ **[F6-Zoom Out]**

Assigning Constant Head Boundary Conditions

The next step is to assign Constant Head boundary conditions to the confined and unconfined aquifers at the north and south boundaries of the model.

As an alternative to clicking **Boundaries/Constant Head** from the top menu bar, use the overlay picklist on the left-hand toolbar to switch from the **Recharge** overlay to the **Const. Head** overlay (scroll up the list to find **Const. Head**).

☞ **[Yes]** to save your Recharge data

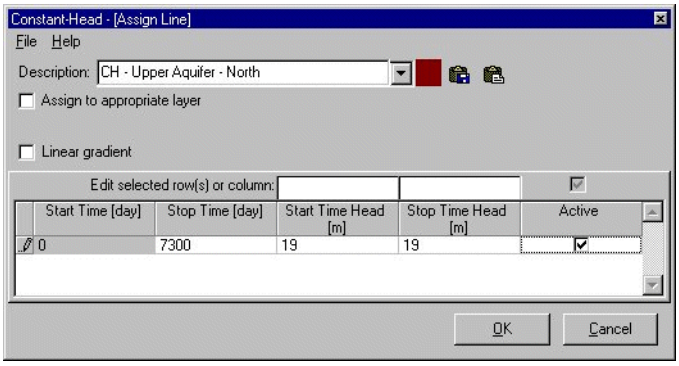
The first Constant Head boundary condition to assign will be for the upper unconfined aquifer along the northern boundary of the model domain. To do this you will use the **[Assign >] Line** tool.

☞ **[Assign >] Line** from the left hand toolbar

Move the mouse pointer to the north-west corner of the grid (top-left grid cell) and click on this location to anchor the starting point of the line.

Now move the mouse pointer to the north-east corner of the grid (top-right grid cell) and **Right-Click** on this location to indicate the end point of the line.

The horizontal line of cells will be highlighted and a **Constant - Head - [Assign Line]** window will appear as shown in the following figure:



Type the following values:

Description:	**CH - Upper Aquifer - North**
Stop Time (day):	**7300** ⇔
Start Time Head (m):	**19** ⇔
Stop Time Head (m):	**19**

☞ **[OK]** to accept these values

The pink line of grid cells will now turn to a dark red color indicating that a Constant Head boundary condition has been assigned to these cells.

Next you will assign a Constant Head boundary condition along the northern boundary for the lower confined aquifer.

☞ **[Goto]** on the left toolbar

The **Go To Layer** window will appear. In the **Layer you wish to go to** field, the number 1 will be highlighted as the default setting.

Type: 3

☞ **[OK]** to advance to layer 3

Check the Navigation Cube to ensure that you are viewing Layer 3. The current layer number is displayed next the field labelled **Layer (K)**.

Now assign the Constant Head boundary condition to the line of grid cells along the northern boundary of Layer 3.

☞ **[Assign >] Line**

Move the mouse pointer to the north-west corner of the grid (top-left grid cell) and click on this location to anchor the starting point of the line.

Now move the mouse pointer to the north-east corner of the grid (top-right grid cell) and **Right-Click** on this location to indicate the end point of the line.

A horizontal line of cells will be highlighted and a **Constant- Head - [Assign Line]** window will appear.

Type the following values:

Description:	**CH - Lower Aquifer - North** ⇔
Stop Time (day):	**7300** ⇔
Start Time Head (m):	**18** ⇔
Stop Time Head (m):	**18**

☞ **[OK]** to accept these values

The pink line of grid cells will now turn to a dark red color indicating that a Constant Head boundary condition has been assigned to these cells.

Next, assign the Constant Head boundary condition to the lower confined aquifer along the southern boundary of the model domain.

☞ **[Assign >] Line**

Move the mouse pointer to the south-west corner of the grid (bottom-left grid cell) and click on this location to anchor the starting point of the line.

Now move the mouse pointer to the south-east corner of the grid (bottom-right grid cell) and **Right-Click** on this location to indicate the end point of the line.

A horizontal line of cells will be highlighted and a **Constant- Head - [Assign Line]** window will appear.

Type the following values:

Code description:	**CH - Lower Aquifer - South** ⇔
Stop time (day):	**7300** ⇔
Start Time Head (m):	**16.5** ⇔
Stop Time Head (m):	**16.5**

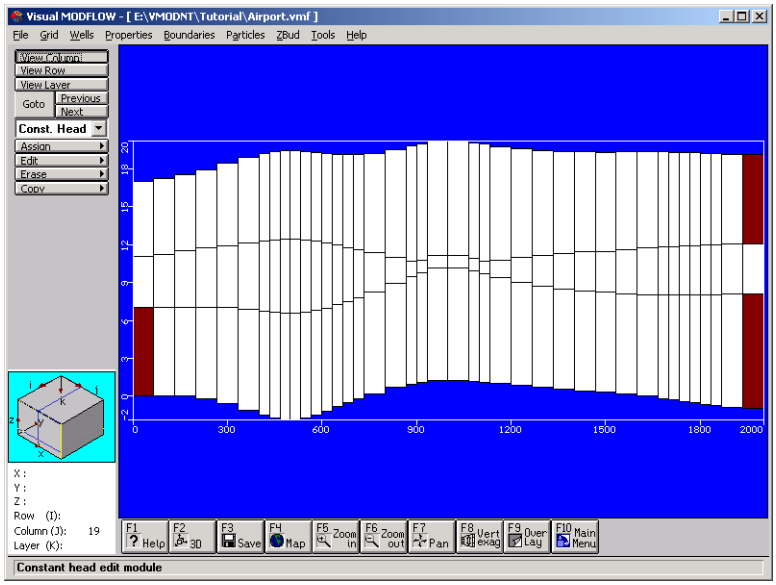
☞ **[OK]** to accept these values

The pink line of grid cells will now turn to a dark red color indicating that a Constant Head boundary condition has been assigned to these cells.

Now view the model in cross-section along a column to see where the Constant Head boundary conditions have been assigned.

☞ **[View Column]** from the left toolbar

Move the mouse into the model domain and select the column passing through the zone of discontinuous aquitard. A cross-section similar to that shown in the following figure should be displayed (viewing column 19).



The hydraulic conductivity property zones can also be displayed at the same time.

☞ **[F9 - Overlay]**

The **Overlay Control** window will appear providing a list of available overlays that may be turned on and off.

Use the **Overlay List Filter** to display only the **Properties** overlays by clicking the button and selecting **Properties**.

- ☞ **☑ Prop(F) - Conductivities**
- ☞ **[OK]** to display the Conductivity overlay

The grid cells in the middle layer of the model (Layer 2) should be colored dark blue, except in the region of the Discontinuous Aquitard, to indicate they belong to Conductivity Zone #2.

Now return to the layer view of the model domain.

- ☞ **[View Layer]** button from the left toolbar

Move the mouse into the cross-section and select Layer 1.

Assigning River Boundary Conditions

If you look closely at the sitemap, you will see the Waterloo River flowing along the southern portion of the model domain (drawn in a light blue color). The following instructions describe how to assign a River boundary condition in the top layer of the model.

- ☞ **Boundaries/River** from the top menu bar to enter the River input screen
- ☞ **[Assign >] Line** from the left toolbar

Beginning on the south-west side of the grid and using the sitemap as a guide, digitize a line of grid cells along the river by clicking along its path with the left mouse button.

When you have reached the south-east boundary, **Right-Click** the mouse button at the end point of the line.

The grid cells along the Waterloo River will be highlighted in pink and **River - [Assign Line]** window (as shown in the following figure) will prompt you for information about the river.

Traditionally, the River boundary condition has required a value for the Conductance of the riverbed. However, the Conductance value for each grid cell depends on the length and width of the river as it passes through each grid cell. Therefore, in a model such as this, with different sizes of grid cells, the Conductance value will change depending on the size of the grid cell.

In order to accommodate this type of scenario, Visual MODFLOW allows you to enter the actual physical dimensions of the river at the Start point and End point of the line, and then calculates the appropriate Conductance value for each grid cell according to the standard formula.

Type the following information for the River boundary condition:

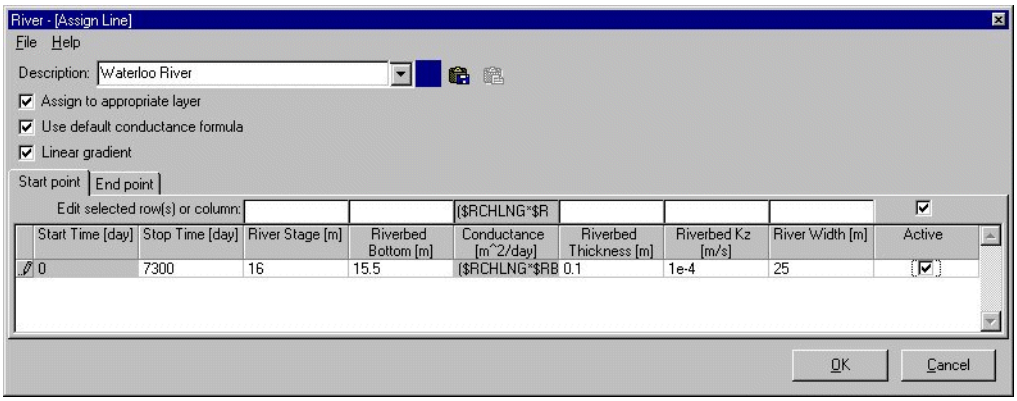
Description: Waterloo River

- ☞ **☑.Assign to appropriate layer**
- ☞ **☑.Use default conductance formula**
- ☞ **☑.Linear gradient**

Note that when the checkbox beside the **Linear gradient** is checked, two tabs, **Start Point** and **End Point** appear in the window (as shown in the following figure)

- ☞ **Start Point** tab and **Type** the following information:

Stop time (day): **7300** ⇔
River Stage Elevation (m): **16.0** ⇔
River Bottom Elevation (m): **15.5** ⇔
Riverbed Thickness (m): **0.1** ⇔
Riverbed Kz (m/sec): **1e-4** ⇔
River Width (m): **25**

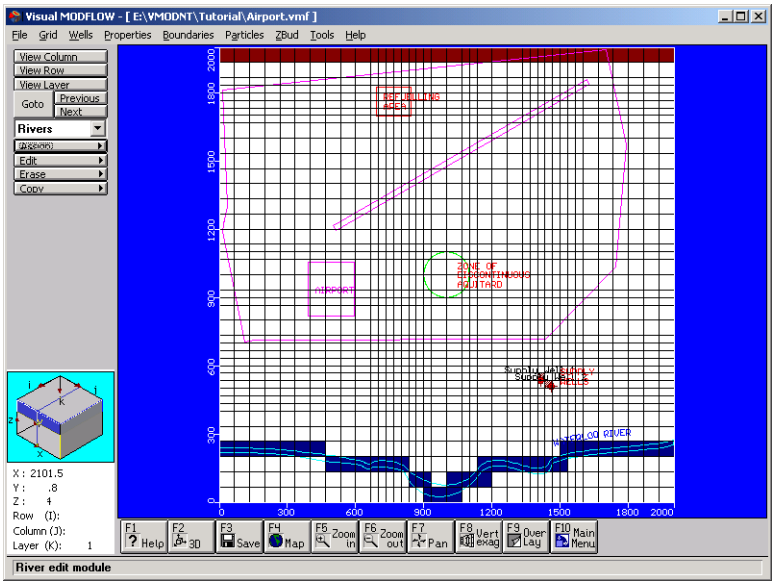


☞ **End Point** tab and **Type** the following information:

Stop time (day): **7300** ⇔
River Stage Elevation (m): **15.5** ⇔
River Bottom Elevation (m): **15.0** ⇔
Riverbed Thickness (m): **0.1** ⇔
Riverbed Kz (m/sec): **1e-4** ⇔
River Width (m): **25**

☞ **[OK]** to accept these values.

After the River boundary condition has been defined, the River grid cells will be colored dark blue, as shown in the following figure:



Section 6: Particle Tracking

Particle tracking is used to determine the preferred flow pathways for a groundwater ‘particle’ starting from a known location. In this section, you will be guided through the steps necessary to assign forward tracking particles to determine the preferred contaminant exposure pathways.

☞ **Particles** from the top menu bar

You will be asked to save the Boundary data,

☞ **[Yes]**

The **Particle** input screen will appear, with the various options for assigning particles displayed along the left-hand toolbar.

For this model you will assign a line of particles in the vicinity of the Refueling Area to determine where the contamination from this location will travel once it is in the groundwater

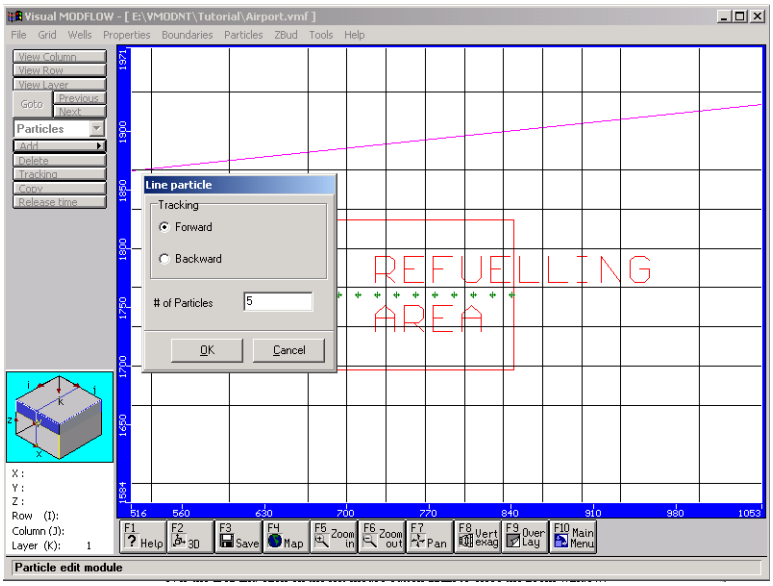
The first step is to once again zoom-in on the Refuelling Area.

☞ **[F5 - Zoom In]** from the bottom toolbar

Click the left mouse button near the top left corner of the Refueling Area, then drag a window over the area and click on the left mouse button again to close the zoom window.

☞ **[Add>] Add Line** to add a line of particles in Layer 1

Move the cursor to the middle left side of the refuelling area and click the left mouse button. Stretch a line to the right side of the refuelling area then click again. A **Line Particle** window will appear, as shown in the following figure:



The default number of particles assigned is **10**. Change the number of particles to **5**.

# of Particles: 5 ↵

☞ **[OK]**

The line of green particles through the refuelling area indicates forward tracking particles. Now return to the full-screen display of the model domain.

☞ **[F6 - Zoom Out]** (from the bottom toolbar)

At this point the first Module of this tutorial has been completed and you have now successfully built a three-dimensional groundwater flow and particle tracking model.

The next Module of this tutorial deals with setting up the contaminant transport model, and assigning the properties and boundary conditions for the transport model.

Note: You can skip Module II: The Contaminant Transport Model on page 41 and proceed to “Module III: Running Visual MODFLOW on page 45 if you want to work only with the Flow model.

Module II: The Contaminant Transport Model

Module II provides instructions on setting up the Transport Model and assigning the required properties and boundary conditions for the contaminant transport model.

If you have skipped MODULE I, then you may start at this point by opening the **Airport2.vmf** project located in the **Tutorial** folder of the Visual MODFLOW installation directory.

Section 7: Setting Up the Transport Model

The following section outlines the steps necessary to complete a simplified transport model. For the purposes of this tutorial the focus will be on the basic setup using MT3DMS, the public domain contaminant transport program.

Contaminant Transport Properties

In this example, the only reaction that will be simulated is linear equilibrium adsorption of a single dissolved species (the diesel fuel compound **JP-4**). However, depending on the complexity of the problem you are dealing with, your model may be required to have several different zones with different sorptive and reactive properties (distribution coefficients, decay coefficients and yield coefficients) for each dissolved species in the model.

In Visual MODFLOW, the properties and processes for the transport model are assigned using the same types of graphical tools as you used for assigning the flow model properties.

Viewing Sorption Properties

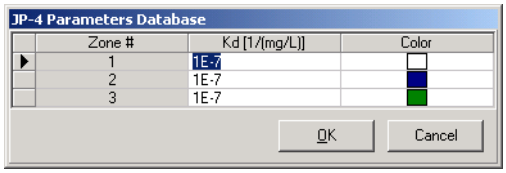
For this tutorial you will not need to modify the Distribution Coefficient value you defined during the transport model setup, but you may examine the sorption parameter values as follows.

- ☞ **Properties/Species Parameters** from the top menu bar

You will be prompted to save changes,

- ☞ **[Yes]**
- ☞ **[Database]** from the left toolbar

The **JP-4 Parameters Database** window will appear as shown below.



The default distribution coefficient (**Kd = 1.0e-7** L/mg) was specified during the setup of the transport numeric engine. If this is not so (e.g. if you did not enter this value when creating the project), enter them now. Note that three separate zones were created, one for each layer of the model.

The Kd values for each zone can be modified to accommodate heterogeneous soil properties and reactions throughout the model domain. However, for this example you will keep it simple and use a uniform Distribution Coefficient for each layer of the model.

☞ **[OK]** to accept the current settings

Assigning Dispersion Coefficients

The next step is to define the dispersion properties for the model.

☞ **Properties/Dispersion** from the top menu bar

You will be asked to save your data.

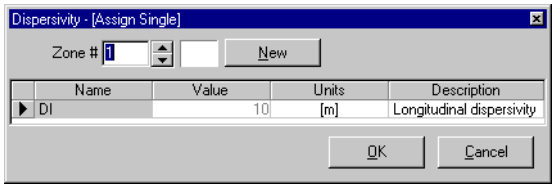
☞ **[Yes]**

Visual MODFLOW automatically assigns a set of default values for each of the dispersivity variables. The following table summarizes these values.

Longitudinal Dispersivity	10
Horizontal to Longitudinal Ratio	0.1
Vertical to Longitudinal Ratio	0.01
Molecular Diffusion Coefficient	0.0

It is possible to assign alternate values for the longitudinal dispersivity by using the **[Assign>]** option buttons from the left toolbar.

☞ **[Assign>] > Single**



The **Dispersivity [Assign Single]** window with default dispersivity value will be displayed. If you have spatially variable dispersion at your site, you can create different zones of grid cells containing unique dispersion values. However, for this example, you will use a uniform dispersion value for the entire model domain.

☞ **[OK]** to accept the default value

In order to modify the horizontal or vertical dispersivity ratios and/or the molecular diffusion values you can use the **[Layer Options]** button from the left toolbar. MT3D only allows the dispersivity ratios to be defined on a layer-by-layer basis - they cannot be defined on a cell-by-cell basis.

For this example you will not need to modify the default dispersivity ratios.

☞ **[OK]** to accept the default value

Section 8: Transport Boundary Conditions

In this section you will define the location and concentration of the contaminant source. The source of contamination will be designated at the refuelling area as a Recharge Concentration that serves as a source of contamination to infiltrating precipitation.

☞ **Boundaries/ Recharge Concentration** from the top menu bar

You will be prompted to save your data.

☞ **[Yes]**

Check to ensure that you are viewing Layer 1.

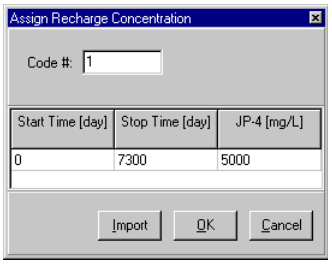
Notice the dark blue grid cells near the Refuelling Area. These grid cells indicate where Recharge Zone #2 is located. The Recharge Concentration input screen automatically displays the different aquifer Recharge zones to help you assign the Recharge Concentration values in the appropriate areas.

Once again, zoom in on the area of interest at the Refuelling Area.

☞ **[F5 - Zoom In]** and stretch a zoom window over the refuelling area

Next, assign the concentration to the natural recharge entering the model at the refuelling area:

☞ **[Assign >] Polygon** from the left toolbar



Move the mouse around the perimeter of the Refuelling Area, clicking on the corners of the area with the left mouse button, then **Right-Click** to close the polygon. The area will become shaded in a pink color, and the **Assign Recharge Concentration** window will appear prompting you to **Type** the following:

Code #: **1**
Stop Time (day): **7300** ⇔
JP-4 (mg/L): **5000**

☞ **[OK]** to accept these recharge concentration values.

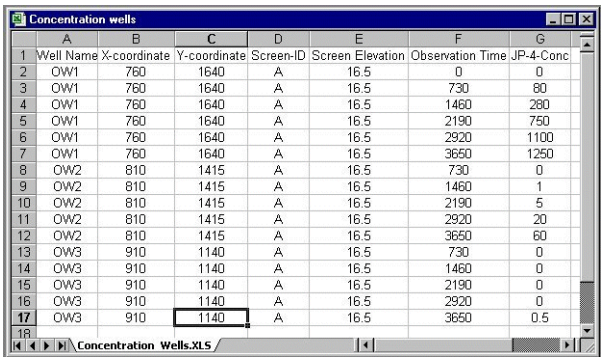
☞ **[F6 - Zoom Out]** to return to the full-screen view of the model domain

Section 9: Adding Observation Wells

In the final step, before running the simulation, you will add the three observation wells to the model to monitor the JP-4 jet fuel concentrations at selected locations down-gradient of the Refueling Area. The first observation well (OW1) was installed immediately down-gradient of the Refuelling Area shortly after the refuelling operation started. The other two observation wells (OW2 and OW3) were installed two years later when elevated JP-4 concentrations were observed at the first well (OW1).

You can add the Concentration Observation Wells individually using the toolbar options on the left-hand menu, or you may import well data from a variety of file formats including ASCII (.TXT), Excel (.XLS), ESRI (.SHP), or MS Access (.MDB). In this section, we will

import concentration observation wells data from an Excel spreadsheet, as shown in the following figure:



	A	B	C	D	E	F	G
1	Well Name	X-coordinate	Y-coordinate	Screen-ID	Screen Elevation	Observation Time	JP-4-Conc
2	OW1	760	1640	A	16.5	0	0
3	OW1	760	1640	A	16.5	730	80
4	OW1	760	1640	A	16.5	1460	280
5	OW1	760	1640	A	16.5	2190	750
6	OW1	760	1640	A	16.5	2920	1100
7	OW1	760	1640	A	16.5	3650	1250
8	OW2	810	1415	A	16.5	730	0
9	OW2	810	1415	A	16.5	1460	1
10	OW2	810	1415	A	16.5	2190	5
11	OW2	810	1415	A	16.5	2920	20
12	OW2	810	1415	A	16.5	3650	60
13	OW3	910	1140	A	16.5	730	0
14	OW3	910	1140	A	16.5	1460	0
15	OW3	910	1140	A	16.5	2190	0
16	OW3	910	1140	A	16.5	2920	0
17	OW3	910	1140	A	16.5	3650	0.5
18							

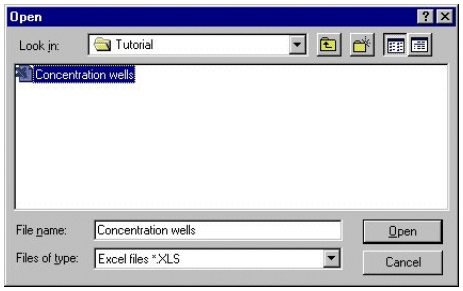
☞ **Wells/Conc. Observation Wells** from the top menu bar

You will be prompted to save your data.

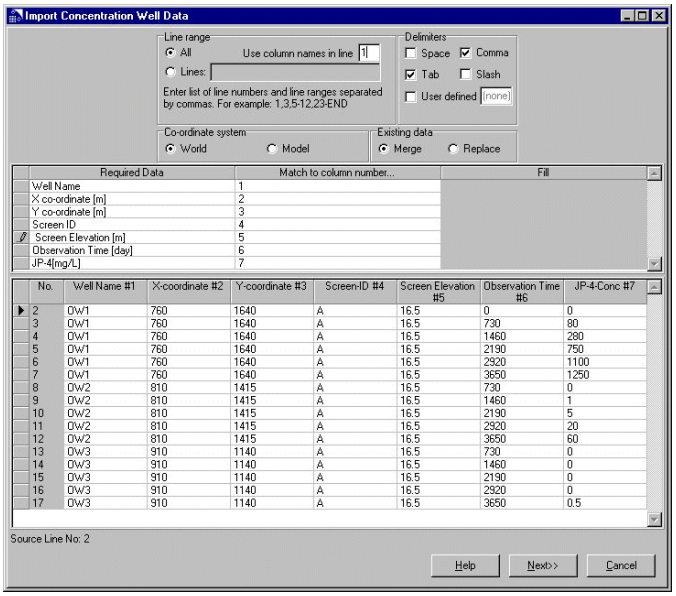
☞ **[Yes]**

☞ **[Import Obs.]** from the left toolbar

The **Open** input window will appear as shown below. Change the **Files of type:** field to **Excel files *.XLS**, and browse for the data file **Concentration wells.XLS**.



☞ **[Open]** to open the following window.



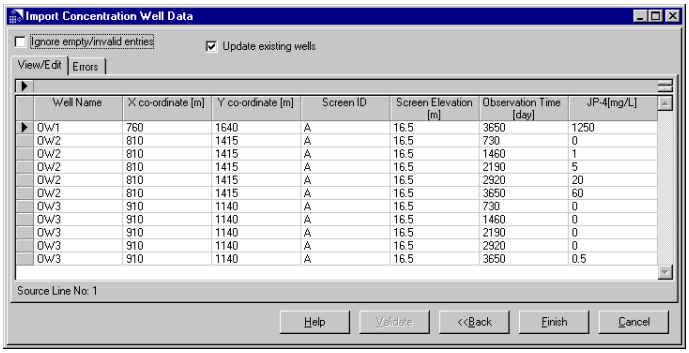
	No.	Well Name #1	X-coordinate #2	Y-coordinate #3	Screen-ID #4	Screen Elevation #5	Observation Time #6	JP-4-Conc #7
1	2	OW1	760	1640	A	16.5	0	0
2	3	OW1	760	1640	A	16.5	730	80
3	4	OW1	760	1640	A	16.5	1460	280
4	5	OW1	760	1640	A	16.5	2190	750
5	6	OW1	760	1640	A	16.5	2920	1100
6	7	OW1	760	1640	A	16.5	3650	1250
7	8	OW2	810	1415	A	16.5	730	0
8	9	OW2	810	1415	A	16.5	1460	1
9	10	OW2	810	1415	A	16.5	2190	5
10	11	OW2	810	1415	A	16.5	2920	20
11	12	OW2	810	1415	A	16.5	3650	60
12	13	OW3	910	1140	A	16.5	730	0
13	14	OW3	910	1140	A	16.5	1460	0
14	15	OW3	910	1140	A	16.5	2190	0
15	16	OW3	910	1140	A	16.5	2920	0
16	17	OW3	910	1140	A	16.5	3650	0.5

In this window,

Type: 1 in the **Use column names in line** field

Match the column numbers to the corresponding fields. When you are finished,

☞ **[Next]** and the following window will appear.


Import Concentration Well Data
Ignore empty/invalid entries
Update existing wells
View/Edit
Errors
Well Name
X co-ordinate [m]
Y co-ordinate [m]
Screen ID
Screen Elevation [m]
Observation Time [day]
JP:4[mg/L]
OW1 760 1640 A 16.5 3650 1250
OW2 810 1415 A 16.5 730 0
OW2 810 1415 A 16.5 1460 1
OW2 810 1415 A 16.5 2190 5
OW2 810 1415 A 16.5 2920 20
OW2 810 1415 A 16.5 3650 60
OW3 910 1140 A 16.5 730 0
OW3 910 1140 A 16.5 1460 0
OW3 910 1140 A 16.5 2190 0
OW3 910 1140 A 16.5 2920 0
OW3 910 1140 A 16.5 3650 0.5
Source Line No: 1
Help
Validate
<<Back
Finish
Cancel

☞ **[Finish]** to import the wells.

When you are finished assigning Concentration Observation wells, click the **[F6-Zoom Out]** button in the bottom toolbar to refresh the screen.

To prepare for the next Module, select **[F10 - Main Menu]** from the bottom toolbar and select **[Yes]** to save your data.

Module III: Running Visual MODFLOW

Module III guides you through the selection of some of the MODFLOW, MODPATH and MT3D run options that are available with Visual MODFLOW.

> If you have skipped Module I & II, then you may start at this point by opening the **Airport3.vmf** project located in the **Tutorial** folder of the Visual MODFLOW installation directory.

To continue with the run options,

☞ **[Run]** from the Main Menu options

Section 10: MODFLOW Run Options

Prior to running the simulation, you may customize the default run-time settings for MODFLOW, MODPATH and MT3D. The Run section also allows you to setup the parameter estimation (PEST) simulation.

The MODFLOW Run-Time options allow you to customize the default settings for various MODFLOW settings including:

- **Time-steps** in each Stress Period (activated only when transient flow is simulated)
- **Initial Heads** estimate
- **Solver** selection and settings
- **Recharge** settings
- **Layers** types
- Dry cell **Rewetting** settings
- **Anisotropy** settings

- **Output Control** setting

MODFLOW Solver Options

First you will examine the Solver options for MODFLOW.

☞ **MODFLOW-2005/Solver** from the top menu bar

The default solver is the **WHS Solver**, a proprietary solver developed by Schlumberger Water Services. To select a different solver, simply click the radio button next to the acronym of the desired solver and the associated solver parameters will be displayed.

For MODFLOW-SURFACT flow models, Visual MODFLOW supports the powerful PCG5 solver, which includes more options for decomposition of the matrix and reduced system ordering schemes. This solver has been proven to be 20 times faster than the PCG4 solver due to its advanced computational techniques. Please refer to the Visual MODFLOW User's Manual for more information.

For this example, the **WHS Solver** will be used to calculate the flow solution.

☞ **[OK]** to accept the WHS Solver

Dry Cell Wetting Options

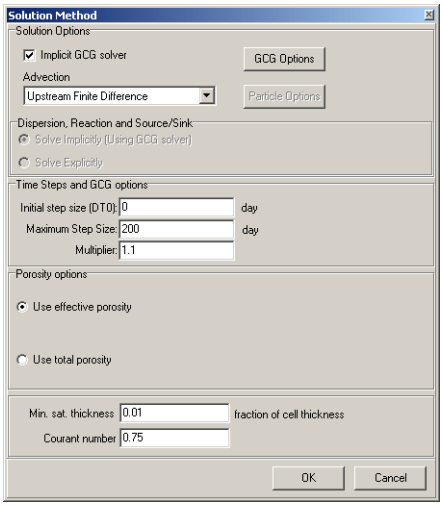
During flow simulations it is common for cells to become dry when the water table drops below the bottom of the grid cell. The dry cell thus becomes inactive and will NOT be able to 're-wet' unless the cell wetting option is activated. By default, the cell wetting option is not active when a new model is created. Therefore, the cell wetting option must be manually activated as follows:

☞ **MODFLOW 2005/ Rewetting**

The **Dry Cell Wetting Options** window will appear.

☞ **☑ Activate cell wetting**

The remaining default settings for the **Dry Cell Wetting Options** (as shown in the figure on the right) are fine for this model.



Note the addition of two non-standard options at the bottom of this window. In instances where you are using a parameter estimation program like WinPEST, the automated parameter adjustments that take place during the iterations may cause dry cells to occur where observations are encountered. Typically, this will confuse the parameter estimation program because MODFLOW assigns a calculated head value of 1e-30 for the dry cell. However, Visual MODFLOW gives you the option to assign the head value for dry cells as the cell bottom elevation.

Visual MODFLOW also gives you the option to prevent dry cells from occurring on the bottom layer of the model.

☞ **[OK]** to accept the remaining Cell wetting options

There are several other MODFLOW run options available for flow simulations. Feel free to investigate the options that were not mentioned in this tutorial.

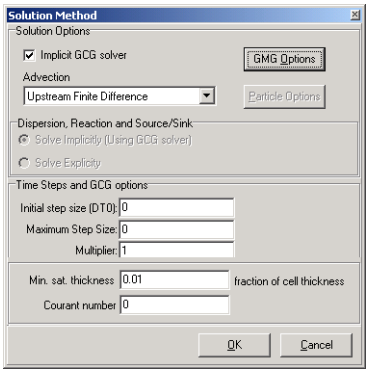
Section 11: MT3DMS Run Options

This section will guide you through the selection of the advection method, solver settings, and output times that you will use to obtain the solution and results for the contaminant transport model.

MT3DMS Solver Settings

☞ **MT3DMS/Solution Method** from the top menu bar

A **Solution Method** window will appear, as shown in the following figure.



In the **Solution Method** dialogue**, Advection term** frame, you will be using the **Upstream Finite Difference** solution method with the Generalized Conjugate Gradient (GCG) solver. The Upstream Finite Difference method provides a stable solution to the contaminant transport model in a relatively short period of time. The GCG solver uses an implicit approach to solving the finite difference equations, and is usually much faster than the explicit solution method.

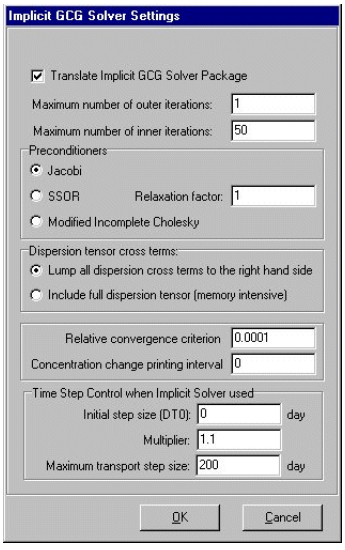
Click the button in the **Advection term** frame, and select **Upstream Finite Difference**

To select the **GCG** Solver

☞ **☑ Implicit GCG Solver**

☞ **[GCG Options]**

The **Implicit GCG Solver Settings** window will appear, as shown in the following figure:


Implicit GCG Solver Settings
Translate Implicit GCG Solver Package
Maximum number of outer iterations: 1
Maximum number of inner iterations: 50
Preconditioners
Jacobi
SSOR
Relaxation factor: 1
Modified Incomplete Cholesky
Dispersion tensor cross terms:
Lump all dispersion cross terms to the right hand side
Include full dispersion tensor (memory intensive)
Relative convergence criterion 0.0001
Concentration change printing interval 0
Time Step Control when Implicit Solver used
Initial step size (DT0): 0 day
Multiplier: 1.1
Maximum transport step size: 200 day
OK
Cancel

Though the Upstream Finite Difference method and the Implicit GCG Solver are computationally efficient, the tutorial simulation tracks contaminant transport over a 20 year period. In order to speed up the modeling process, you will use a non-linear time step.

Type the following information in the fields at the bottom of the window.

Multiplier: 1.1

Maximum transport step size: 200

☞ **[OK]**

This will return you to the **Solution Method** window.

Type the following values for the Minimum Saturated Thickness and Courant Number:

Min. sat. thickness: 1E-2

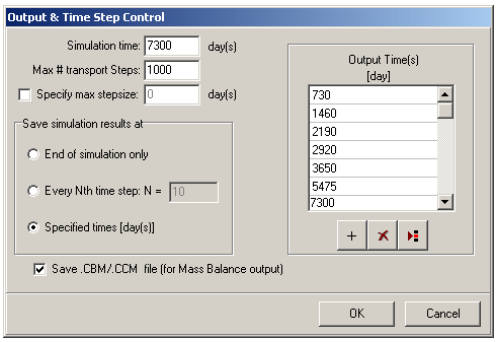
Cournat number: 7.5E-1

☞ **[OK]**

MT3DMS Output Times

Next, you will define the output times at which you would like to see the contaminant transport modeling results.

☞ **MT3DMS/Output/Time steps** from the top menu bar



An **Output & Time Step Control** window will appear. The default setting for MT3D output control is to save only one set of concentration data values at the **End of simulation only**. For this tutorial you will define Specified times at which you would like to see the simulation results.

Fill in and check off the following:

Simulation time: **7300 days**

Max # of transport steps: **10000**

☞ **⊙ Specified times [days(s)]**

In the **Output Time(s)** list, **Type** the following values:

730 ↵
1460 ↵
2190 ↵
2920 ↵
3650 ↵
5475 ↵
7300

☞ **[OK]** to accept these Output & Time Step Control parameters

This concludes the MT3D run options section of the tutorial. There are many other run settings that have not been covered in this tutorial. Fell free to investigate some of the other options available.

Section 12: Running the Simulation

To run your simulation,

☞ **Run** from the top menu bar

An **Engines to Run** window will appear listing the available numeric engines (as shown in the figure below).

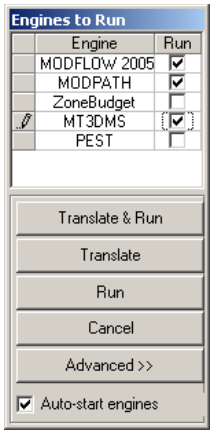
Although the flow model will solve relatively quickly, the contaminant transport simulation will take between 1 and 5 minutes to run to completion depending on the processor speed and amount of RAM on your computer.

If you are using this tutorial for introductory training purposes, you can choose to either run the simulation for the model you have created, or you can continue with this tutorial, without running the model. If you do not have time to run the simulation, the Output data files are provided with the example projects.

To continue with this tutorial without running the model, press **[Cancel]** and select the **[F10 - Main Menu]** button from the bottom toolbar and then proceed from Module IV - Output Visualization on page 49.

To run the simulation follow the instructions below.

In the **Engines to Run** window, select the following options.



- ☞ ☑ **MODFLOW-2005**
- ☞ ☑ **MODPATH**
- ☞ ☑ **MT3DMS**
- ☞ **[Translate & Run]**

Visual MODFLOW will then **Translate** the Visual MODFLOW data set into the standard data input files required for the selected numeric engines, and then **Run** the simulations in a separate window labelled **VM Engines**.

When the engines are finished running, click **[Close]** to close the **VM Engines** window.

MODULE IV: Output Visualization

In this module, you will be visualizing results from your model. If you did not run the Numeric Engines, open the **Airport4.vmf** project located in the **Tutorial** folder of the Visual MODFLOW installation directory.

- ☞ **Output** from the top menu bar of the **Main Menu**

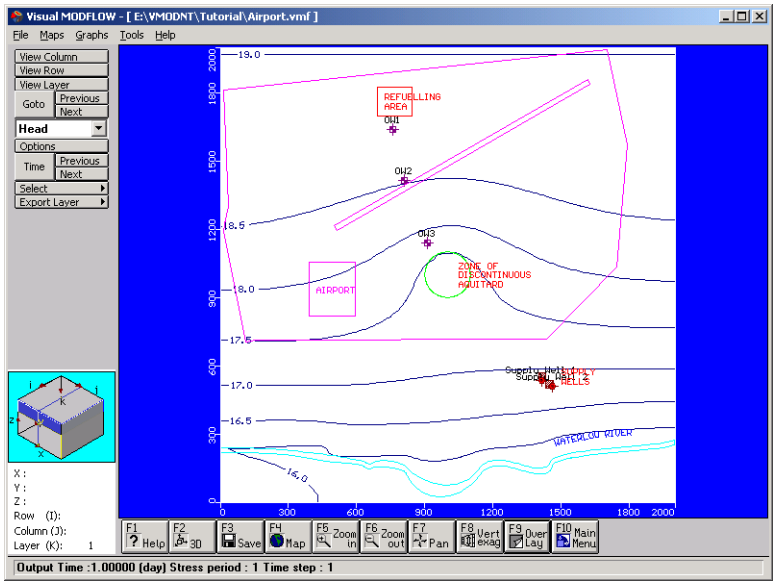
You will be transferred to the Visual MODFLOW Output section where you can visualize, print, and export the simulation results. This Module provides instructions on displaying, interpreting, customizing, and printing the groundwater flow, pathline, and contaminant transport modeling results using a variety of mapping and graphing presentation formats including:

- **Maps:** Contour lines, color maps, pathlines, velocity vectors, calibration bubble plots
- **Graphs:** Calibration scatter graphs, calibration frequency histograms, time series graphs, mass balance graphs, and zone budget graphs

Upon entering the Output section, Visual MODFLOW will automatically load the available Output files for Head (.HDS), and Drawdown (.DDN) for all output times. Once these data files are loaded, the **Output** screen will appear.

The first time the Output section is opened, Visual MODFLOW will display the site map and the head contours for the current layer, row or column (see figure below). When the Output display settings are modified, Visual MODFLOW will always recall the most recent Output display settings.

If the Airphoto map shows up in the output, you may disable it using the **Overlays** dialogue. Your Output window should now appear as follows:

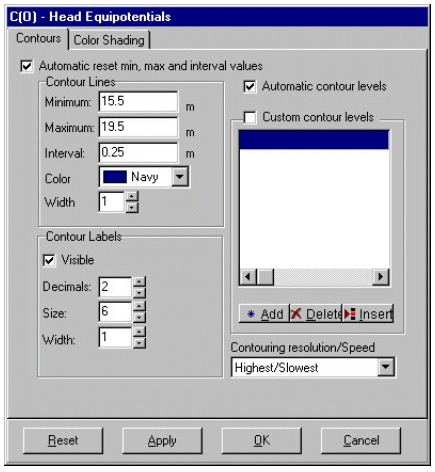

Visual MODFLOW - [E:\VMODNT\Tutorial\Airport.vmf]
File Maps Graphs Tools Help
View Column
View Row
View Layer
Goto Previous Next
Head
Options
Time Previous Next
Select
Export Layer
REFUELLING AREA
OW2
OW3
AIRPORT
ZONE OF DISCONTINUOUS AQUITARD
WATERLOO RIVER
F1 Help F2 3D F3 Save F4 Map F5 Zoom in F6 Zoom out F7 Pan F8 Vert exag F9 Over Lay F10 Main Menu
Output Time :1.00000 (day) Stress period : 1 Time step : 1

Section 13: Head Contours

By default, the contouring algorithm will automatically choose contour settings where the minimum, maximum, and interval values are “attractive”. However, if you do not like the default contour interval or line color, you can customize the contour map settings. To access the contouring options for Head Equipotentials:

☞ **[Options]** from the left toolbar

The **C(O) - Head Equipotentials** window will appear.



Type the following values:

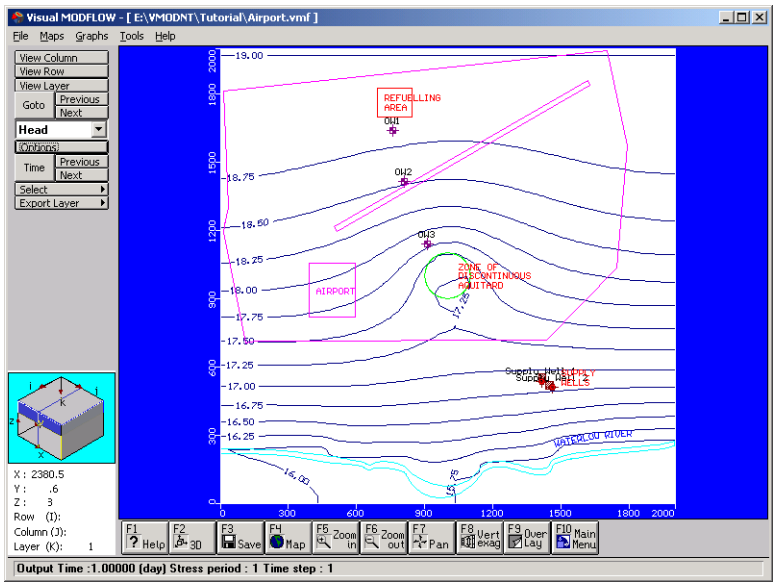
Interval: **0.25**

Decimals: **2**

You can also change the contouring speed and resolution by clicking on the pull-down menu labelled **Contouring resolution/Speed**. For this exercise, the default setting of **Highest/ Slowest** is fine.

☞ **[OK]** to accept these settings

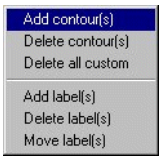
The contour map of the equipotential heads should appear as follows:


Visual MODFLOW - [E:\VMODNT\Tutorial\Airport.vmf]
File Maps Graphs Tools Help
View Column
View Row
View Layer
Goto Previous Next
Head
(Onlayer)
Time Previous Next
Select
Export Layer
REFUELLING AREA
OW2
OW3
AIRPORT
ZONE OF DISCONTINUOUS AQUITARD
WATER COU RIVER
F1 Help F2 3D F3 Save F4 Map F5 Zoom in F6 Zoom out F7 Pan F8 Vert exag F9 Over Lay F10 Main Menu
Output Time :1.00000 [day] Stress period : 1 Time step : 1

All Output contour display screens have additional contour display options available by **Right-Clicking** on the model domain. This will bring up a shortcut menu with options to add, delete, and move contour lines and labels.

Adding Contour Lines

To add some contour lines at selected locations:



Right-Click on the model domain

☞ **Add contour(s)**

Left-Click the mouse pointer anywhere in the model domain to add a contour line for the head value at that location.

Any number of contour lines can be added throughout the model domain by clicking on the desired contour line locations.

When you are finished adding contour lines, **Right-Click** to exit the **Add Contour** mode.

Moving Contour Labels

Next you will move contour line labels.

Right-Click on the model domain

☞ **Move label(s)**

Move the mouse pointer to the location of any contour label and then **Click-and-Drag** the label to a new location on that contour line.

When you are finished moving contour labels, **Right-Click** to exit the **Move Label** mode.

Next, take a look at the site in cross-section.

Select the **[View Column]** button from the left toolbar and move the mouse into the model domain. Choose a column near the middle of the model domain by clicking the left mouse button.

Notice that the minimum, maximum, and interval values for the contour lines are reset each time you change the view, but the custom contour values remain. The minimum, maximum, and interval values are reset to ensure that contour lines are optimized for all data in the current row, column, or layer. However, this reset option can be de-activated in the contour options window if you want to maintain the same maximum, minimum, and interval values for all views.

Deleting Custom Contours

To remove the custom contours,

Right-Click on the model domain

☞ **Delete all custom**

Displaying a Color Map of Heads

Return to the layer view display of the model

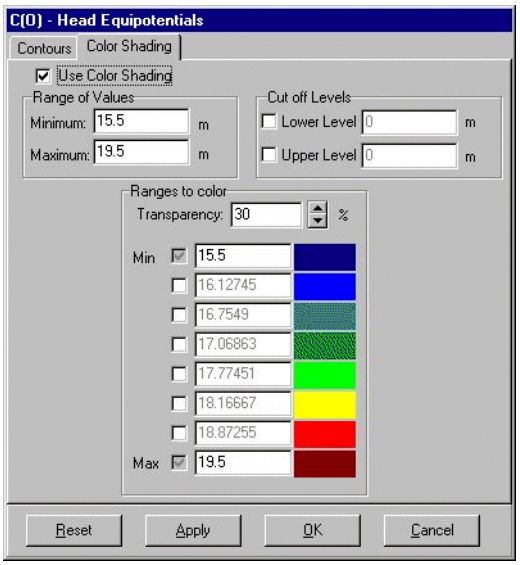
☞ **[View Layer]** button from the left toolbar

Click on Layer 1 in the model cross-section.

A contour map of the heads in Layer 1 will be displayed. To display a color map of the head values in Layer 1:

☞ **[Options]** button from the left-hand tool bar

A **C(O) - Head Equipotentials** contour options window will appear.



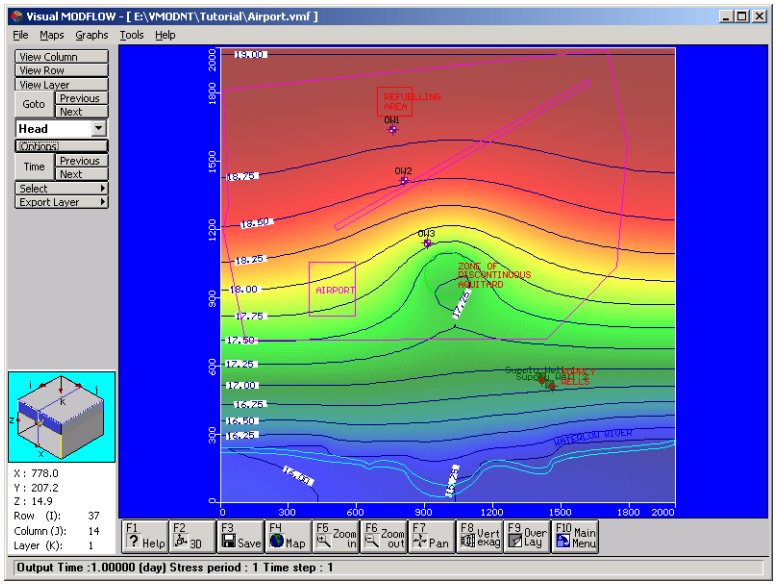
☞ **Color Shading** tab.

The contour options window should appear as shown below.

☞ **☑ Use Color Shading**

☞ **[OK]** to accept these settings

A colormap of the head equipotentials for the top layer of the model will be plotted according to the default color scale selected. In the figure below, note that the color map is semi-transparent in order to show the underlying model features.


Visual MODFLOW - [E:\VMODNT\Tutorial\Airport.vmf]
File Maps Graphs Tools Help
View Column
View Row
View Layer
Goto Previous Next
Head
(Options)
Time Previous Next
Select
Export Layer
X : 778.0
Y : 207.2
Z : 14.9
Row (I): 37
Column (J): 14
Layer (K): 1
F1 Help F2 3D F3 Save F4 Map F5 Zoom in F6 Zoom out F7 Pan F8 Vert exag F9 Over Lay F10 Main Menu
Output Time :1.00000 (day) Stress period : 1 Time step : 1

Before you proceed with this exercise, you should turn-off the color shading option to avoid the long screen refresh times required to redraw the colormap.

- ☞ **[Options]** from the left toolbar
- ☞ **[Reset]** to re-establish the default contour line settings

Click on the **Color Shading** tab.

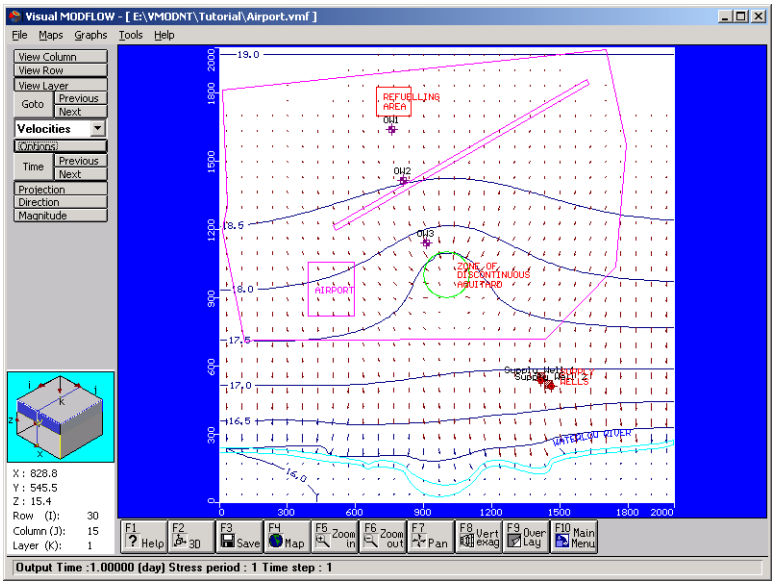
- ☞ Remove the checkmark ✓from the checkbox ☐ beside **Use Color Shading**
- ☞ **[OK]**

Section 14: Velocity Vectors

Velocity vectors are an excellent way to visualize the speed and direction of a water particle as it moves through the flow field. To use this tool:

- ☞ **Maps/Velocities** from the top menu bar to view the velocity vectors

The velocity vectors will be plotted according to the default settings that plot the vectors in relative sizes according to the magnitude of the flow velocity.


Visual MODFLOW - [E:\VMODNT\Tutorial\Airport.vmf]
File Maps Graphs Tools Help
View Column
View Row
View Layer
Goto Previous Next
Velocities
(Options)
Time Previous Next
Projection
Direction
Magnitude
X : 828.8
Y : 545.5
Z : 15.4
Row (I): 30
Column (J): 15
Layer (K): 1
F1 Help F2 3D F3 Save F4 Map F5 Zoom in F6 Zoom out F7 Pan F8 Vert exag F9 Over Lay F10 Main Menu
Output Time :1.00000 (day) Stress period : 1 Time step : 1

By default the color scheme for the velocity vectors is: **red** is outward (i.e. up when viewing by layer), **blue** is inward (i.e. down when viewing by layer), and **green** is parallel to the plane (i.e. horizontal when viewing by layer).

The velocity vectors can also be displayed as directional arrows independent of velocity magnitude.

☞ **[Direction]** from the left toolbar

All of the arrows will then be plotted as the same size, and simply act as flow direction arrows.

☞ **[Options]** from the left menu

Now increase the number of vectors displayed.

Type: Vectors:40 (this specifies the number of velocity vectors along a single row)

☞ ☑ **Autoscale** (automatically adjusts the size of the velocity vectors)

☞ ⊙ **Variable Scale** (prevents overlapping of velocity vectors)

☞ **[OK]** to accept the new velocity vector settings

To see the site in cross-section, select the **[View Column]** button from the left tool bar, and then move the mouse into the model domain and click once to select any column.

☞ **[Projection]** from the left toolbar

To show the different hydraulic conductivity zones, select the **[F9 - Overlay]** button from the bottom menu bar. An **Overlay Control** window will appear with a listing of all the available overlays that you may turn on or off.

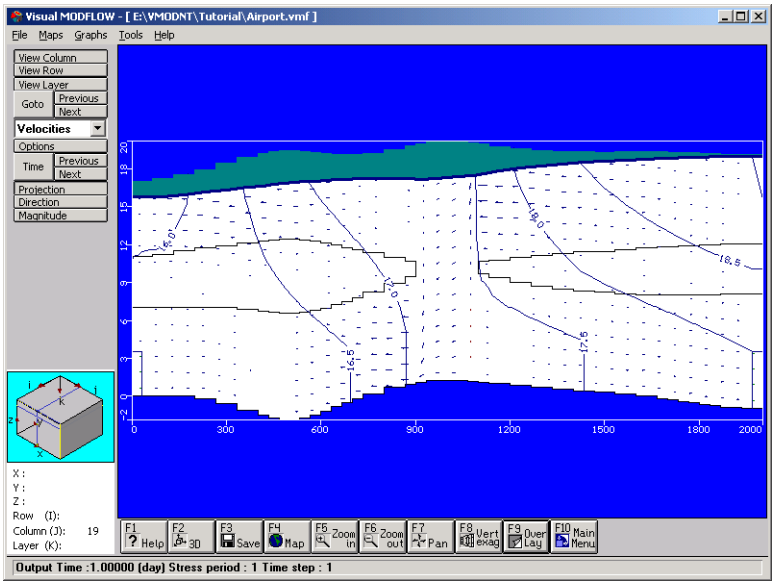
☞ **Overlay List Filter**, and use the ▼ button to select **Properties**

Uncheck **List only visible overlays**

☞ ☑ **Prop(F) - Conductivities**

☞ **[OK]** to show the cross-section with the hydraulic conductivity layers

The display should look similar to the following figure (col. 19).



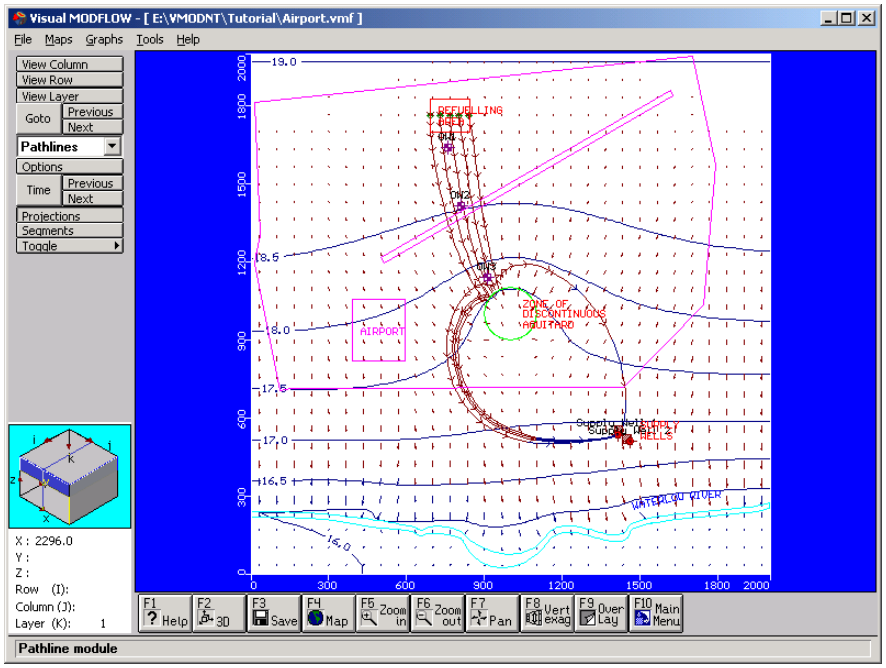
Return to the plan view display of the model by selecting the **[View Layer]** button from the left toolbar and then clicking on Layer 1 in the model cross-section.

Section 15: Pathlines

Recall that in the Input section we assigned five forward tracking particles near the Refueling Area. The MODPATH program tracks the flowpaths of these particles as they travel through the groundwater system. To view the fate of these particles:

☞ **Maps/Pathlines** from the top menu bar

The default screen display will plot all of the pathlines as projections from the current layer of the model (as shown below).



To remove the velocity vectors from the Output display, select the **[F9 - Overlay]** button from the bottom menu bar and select the ☑ **Show only visible overlays** option. The Overlay List will now include only the overlays currently being displayed.

Scan down the listing (towards the middle) and deselect **Vel - Vectors**.

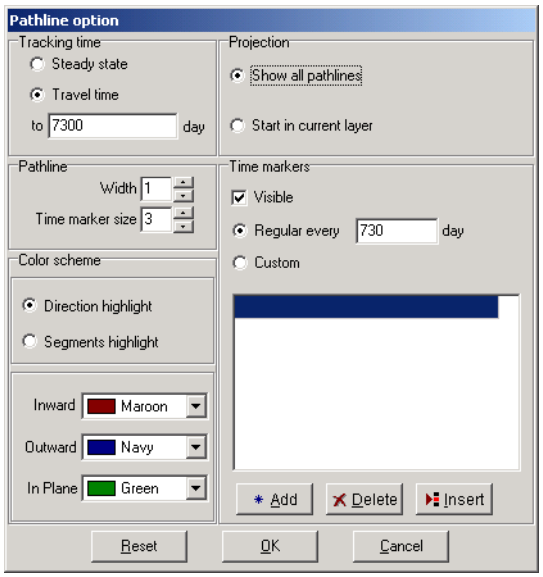
☞ **[OK]** to remove the velocity vectors from the Output display

To show the pathlines in the current layer only, select the **[Segments]** button from the left toolbar. This will show only the pathlines that are in the active layer of the model.

Use the **[Next]** layer button from the left toolbar to advance through each layer of the model to see where the particle pathlines are located.

☞ **[Projections]** to see all pathlines again.

Note that the pathlines have direction arrows on them indicating the flow direction. These arrows also serve as time markers to determine the length of time before a particle reaches a certain destination. To determine the time interval for each time marker, select the **[Options]** button from the left toolbar, and a **Pathline Option** window will appear as shown below:



In some instances, you may want to see how far a contaminant might travel in a given amount of time. For instance, to see how far the particles would travel in 7,300 days;

- ☞ ⊙ **Travel Time** and **Type** a value of **7300**

The **Time markers** should already be selected as ☑ **Visible**.

- ☞ ⊙ **Regular Every** and **Type** a value of **730** in the **day** field
- ☞ **[OK]** to view the time-related pathlines up to a time of 7,300 days

Notice that none of the particles reach the water supply wells after 20 years.

Now view the pathlines in cross-section to see a projection view of the pathlines as they travel through the different layers of the model.

- ☞ **[View Column]** from the left toolbar

Move the cursor across the screen until the highlight bar is in the vicinity of the zone of discontinuous aquitard and click on it.

The cross-sectional view clearly shows all of the pathlines travelling horizontally in the upper aquifer, and then vertically downwards to the lower aquifer through the ‘hole’ in the aquitard.

Return to the plan view display of the model by selecting the **[View Layer]** button from the left toolbar, and then clicking on Layer 1 in the model cross-section.

Section 16: Concentration Contours

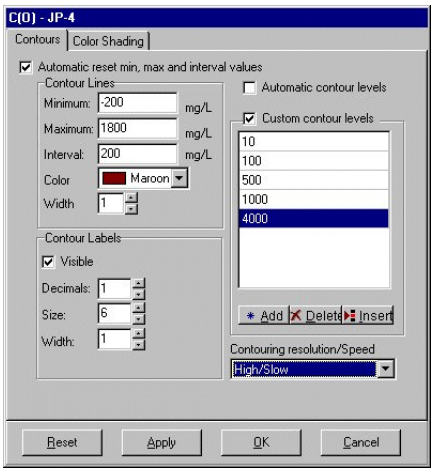
In this section you will learn how to display your contaminant transport simulation results.

- ☞ **Maps/Contouring/Concentration** from the top menu bar

The concentration contours will be plotted for the first transport output time (in this case the first transport output time is 730 days).

At this early time in the simulation, the contaminant plume does not appear to have migrated far from the Refuelling Area. However, this appearance is somewhat misleading because the concentration contours are plotted using a normal scale automatically selected by Visual MODFLOW.

A more effective (and common) method of plotting concentration contours is to plot the contour lines at logarithmic intervals (e.g 0.01, 0.1, 1.0, 10, 100, etc.).



☞ **[Options]** button from the left-hand toolbar.

A Contour Options window for **C(O) - JP-4** will appear, as shown in the figure to the right:

Notice that the minimum concentration value is **-100**. Obviously, a negative concentration value is not possible in a natural environment. This negative concentration value is a result of numerical dispersion encountered during the solution of the contaminant transport simulation.

☞ ☐ **Automatic contour levels** (to remove the automatic contour line)

☞ ☑ **Custom contour levels**

Type the following custom contour levels in the text boxes provided:

10↵
100↵
500↵
1000↵
4000

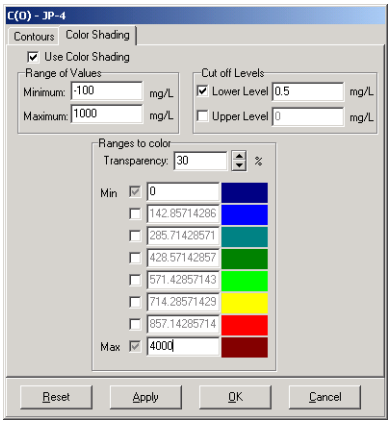
Although this is not a strict logarithmic scale, it does provide a much better representation of the concentration contours.

Next you will increase the drawing speed of the concentration contour lines by changing the **Contouring resolution/speed** setting from **Highest/Slowest** to **High/Slow**.

Finally, you will turn the color shading option on to display a color map of the concentrations.

☞ **Color Shading** tab.

The **Color Shading** options will appear, as shown in the figure to the right:



☞ ☑ **Use Color shading**

In the **Cutoff Levels** section,

☞ ☑ **Lower Level: 10**

The **Lower Level** setting will ‘clip’ the color map at 10 mg/L (all concentration values below 10 mg/L will not be shown on the color map). The **Lower level** setting of 10 mg/L is an action level. If the contamination will be present in the lower aquifer at a value equal to or greater than 10 mg/L, then some action must be performed to remediate the situation.

The **Ranges to color** section lists the range of concentration values for the current Output times. In the **Ranges to Color** section **Type** the following:

Min: 0.0

Max: 4000

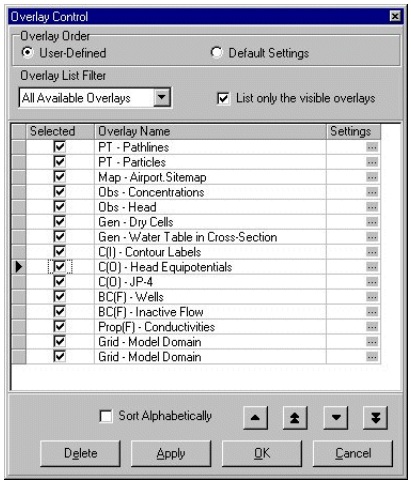
☞ **[OK]**

A maximum concentration value of 4000 was used because this represents the upper limit of the calculated concentrations for the entire simulation time (not just the first Output time of 730 days).

Note that the Concentration Contour Map and Color Map appear transparent in order to show the underlying model features and to display multiple images simultaneously. The reason is that in Visual MODFLOW the drawing tools and algorithms used to render the color shaded contour maps of distribution data have been upgraded to include rich RGB color schemes. However, you may want to modify the default Overlay priority.

☞ **[F9 - Overlay]** from the bottom menu bar

The **List only the visible overlays** option is still selected, so the Overlay List will include only the Overlays currently being displayed.



☞ ⦿ **User Defined**

Scan down the listing (towards the middle) and find **C(O) - Head Equipotentials**.

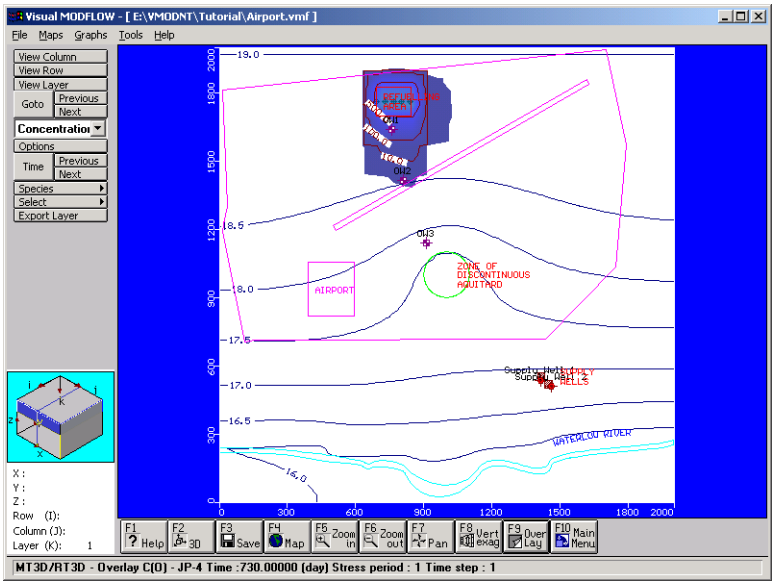
Click on the left-most column (where the black triangle appears in the figure to the right) next to **C(O) - Head Equipotentials** to highlight this row completely in black, then click the **[Up Arrow]** button until it is moved above the **C(O) - JP-4** Overlay (make sure the **C(O) - Head Equipotentials** Overlay is still selected to be displayed).

Finally, you will remove the Pathlines Overlay from the Output display.

☞ ☐ **PT - Pathlines** to disable the Pathlines Overlay

☞ **[OK]**

The Output display should look similar to the following figure.



Place the cursor on the display results, you will see the currently selected **Time: 730** days is reported in the Status Bar (at the bottom of the window).

To see the Concentration Color Map and Contour Map for other Output times.

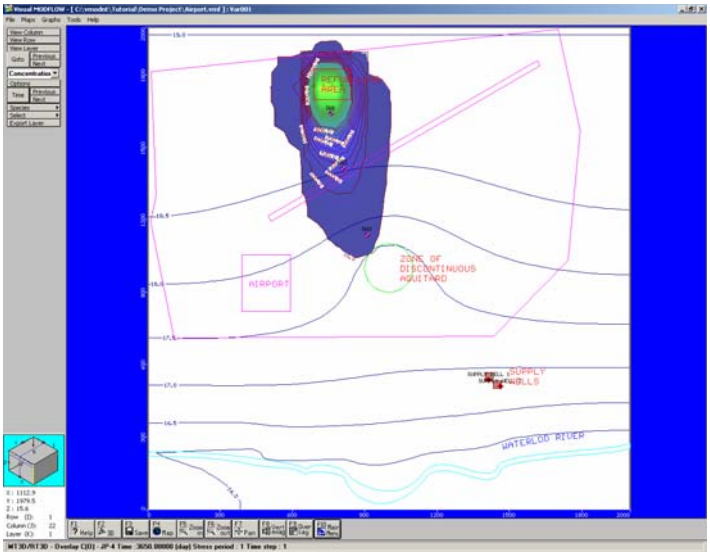
- ☞ **[Time]** from the left toolbar

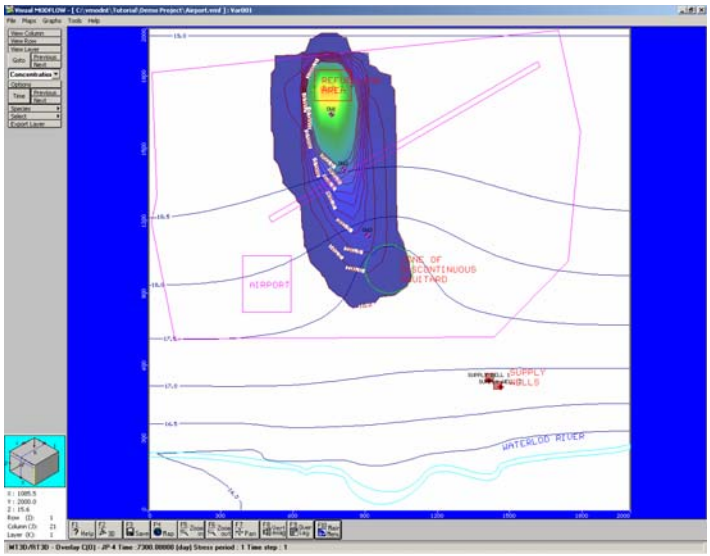
A **Select Output Time** window will appear with a list of the available Output times.

- ☞ **1460**
- ☞ **[OK]**

A plot of concentration contours will appear displaying the results from an Output time of **1460** days. You can advance to the next time step by clicking the **[Next]** button beside the **[Time]** button in the left-hand toolbar.

By **3650** and **7300** days, the concentrations in the first layer of the model should look similar to the following figures:





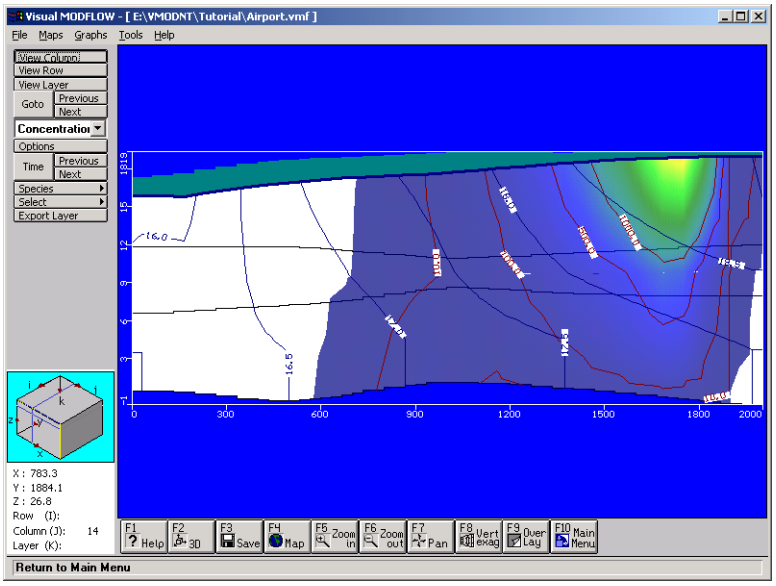
After **3650** days (10 years) of simulation time it is clear that the plume has migrated to the 'hole' in the aquitard.

To see how the plume looks in cross-section,

☞ **[View Column]**

Move the mouse pointer into the model domain and then click on a column passing through the plume.

The cross-section view of the plume should look similar to the following figure.

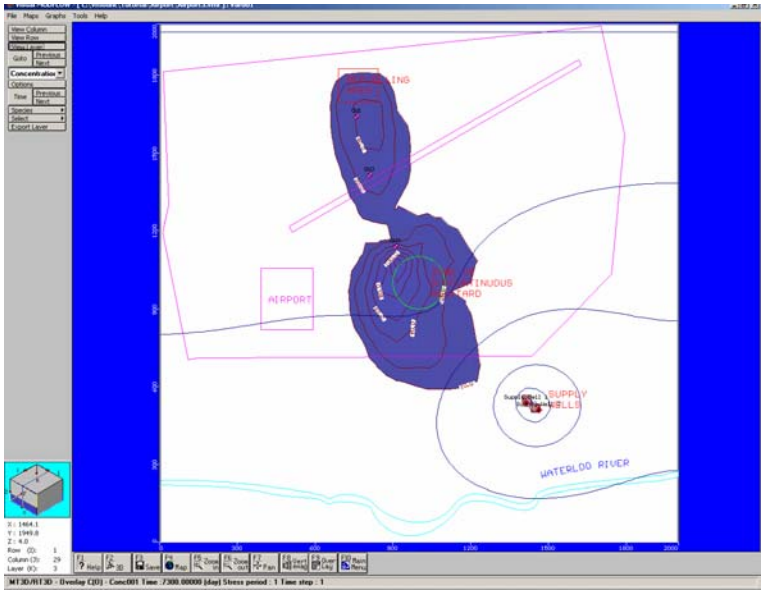


This figure shown the plume has migrated into the lower aquifer at a concentration above the action level and will eventually threaten the pumping wells. To see the plume in the lower aquifer,

☞ **[View Layer]**

Move the mouse pointer into the cross-section view and click on Layer 3.

The display should now look similar to the following figure.



Section 17: Concentration vs. Time Graphs

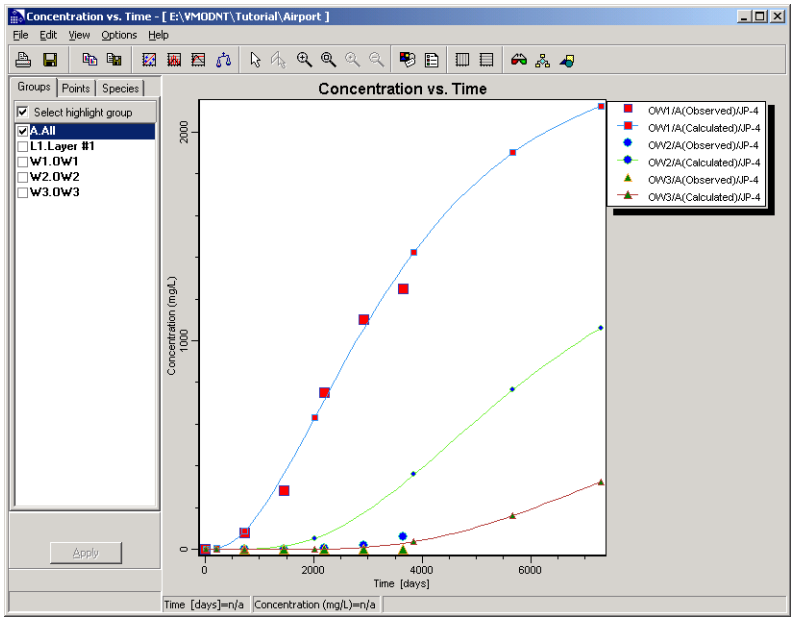
In this section you will learn how to compare the observed concentration data to the concentration values calculated by the model.

- ☞ **Graphs/Time Series/Concentration** from the left toolbar

You will then be presented with the graph for **Concentration vs. Time**, enabling you to view the breakthrough curves at each of your concentration observation wells.

- ☞ **A.All** from the **Groups** box on the left side of the window
- ☞ **[Apply]**

You should now be viewing the breakthrough curves for each of the three concentration observation wells defined earlier in the model (see following figure).



This time-series graph shows the calculated result using a line-and-point data series while the observation data is displayed only as data point symbols. The observation data point symbols are the same color and shape as the calculated values, but the observation data point symbols are slightly larger to make it easier to distinguish them.

This graph can also be presented using a log-scale for the concentrations.

Right-Click on the graph to see a popup menu

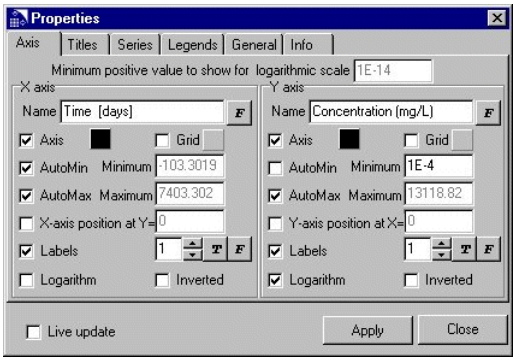
- ☞ **Properties**

A **Properties** window will appear with a comprehensive selection of settings for customizing the appearance and settings for the graph.

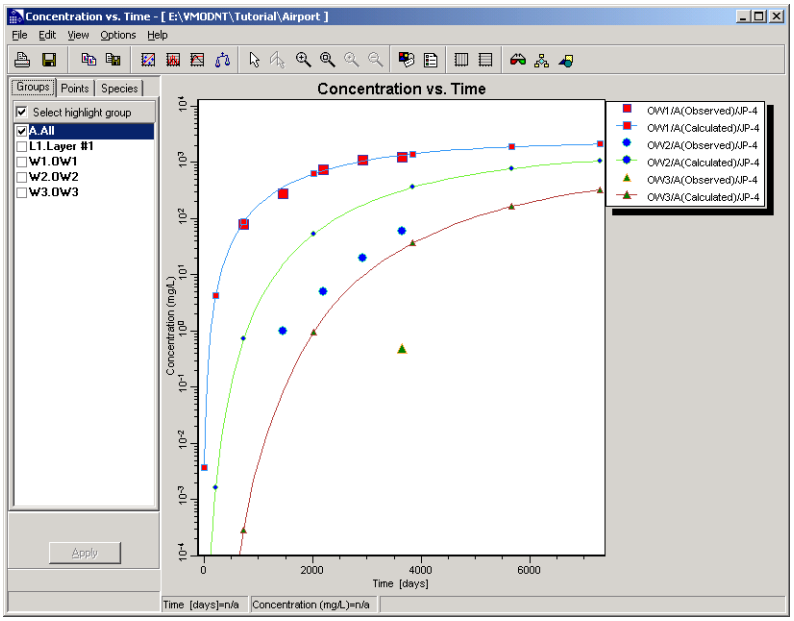
Under the **Y-axis** section,

- ☞ ☑ **Logarithm**
- ☞ ☐ **Automin**
- ☞ **Minimum: 1e-4**

☞ **[Close]**



The **Concentration vs. Time** graph should appear as shown below.



The comparison of the observed vs. calculated concentrations should now be easier to make for the lower concentration values observed at OW2 and OW3.

This graph can be printed, saved to a .BMP, .WMF, .EMF, .DXF, .JPG, .TIF, .GIF, or .PNG format file, or the data can be exported to an ASCII .TXT file, using the **File** options from the top menu bar.

☞ **File>Exit** to close the graph window

Module V: Visual MODFLOW 3D-Explorer

Visual MODFLOW 3D-Explorer is an advanced 3D visualization system for displaying and animating Visual MODFLOW models. Visual MODFLOW 3D-Explorer uses state-of-the-art 3D graphics technology to display your model data and simulation results using a complete selection of graphical formats including:

- 2D and 3D views of the model grid
- Colored shapes for each grid cell
- Bitmap overlays draped over surfaces
- Enhanced contour label positioning and controls

- 3D volumetric shapes for representing model property zones, boundary conditions, and contamination plumes
- Output times synchronized to animate the combined transient flow and transport results
- 3D cutaway views
- Multiple horizontal slices, vertical slices and irregular cross-sections for plotting contour line maps and color maps of modeling results and model properties (heads, concentrations, water table, conductivity values, etc.)
- Concentration plumes, automatically clipped above the water table surface
- Line elements for representing groundwater flow pathlines
- Light source position control
- Transparent objects
- Animation sequence pausing, looping and recording to an AVI file
- Auto-rotate tools to continuously rotate the model image around the X, Y, Z axes
- and much more

The following section will guide you through the process of using Visual MODFLOW 3D-Explorer to display and animate a groundwater flow and mass transport model.

NOTE: 3D-Explorer is only available in the Pro and Premium versions**.**

Section 18: Starting Visual MODFLOW 3D-Explorer

Visual MODFLOW 3D-Explorer can be launched directly from Visual MODFLOW environment, or as a “stand alone” application. In this tutorial you will launch the 3D-Explorer from Visual MODFLOW and visualize the model Input and Output data from the **Airport4.vmf** model.

> The Visual MODFLOW project files for the **Airport4** model are included in the Visual MODFLOW working directory, in the **Tutorial** folder

The 3D-Explorer can be launched from either the Input or Output section of Visual MODFLOW. For this example, you will launch the 3D-Explorer from the Output module.

☞ **Output** from the top menu bar of the **Main Menu**

The 3D-Explorer can now be launched by clicking the **[F2-3D]** icon located on the bottom toolbar.

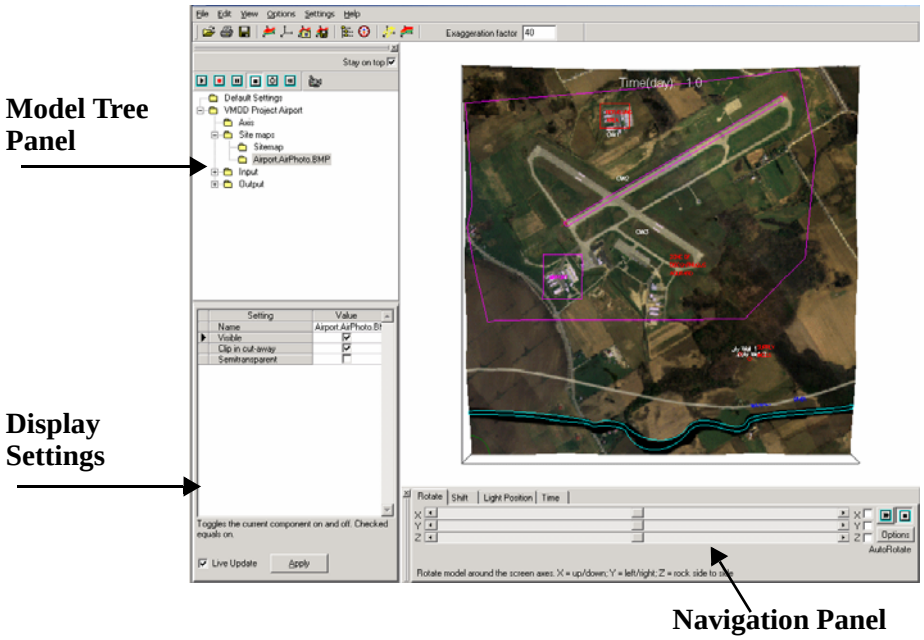
When 3D-Explorer is launched directly from Visual MODFLOW, it will automatically open the active Visual MODFLOW project data files, with the associated sitemap overlays and the vertical exaggeration setting. The first view of the model is a full domain view with the Visual MODFLOW project name at the top of the 3D Display window. When the 3D Explorer window is first opened, it will appear as shown in the following figure.

The **Model Tree Panel** contains a structured listing of the data classes available for the selected Visual MODFLOW project.

The **Display Settings Panel** controls the visual attributes for each display element.

The **Navigation Panel** controls the on-screen orientation and rotation of the 3D graphical images, and the position of the light source.

These will be referred to frequently throughout the remainder of this tutorial.



Section 19: Changing the Default Settings

The **Default Settings** are used to define your preferences for how selected objects are initially displayed when a project is opened. If the **Default Settings** of the selected objects are modified, the new settings may be saved and applied to all VMOD 3D-Explorer projects. For instance, the **Default Setting** for background color is black. To change these default settings,

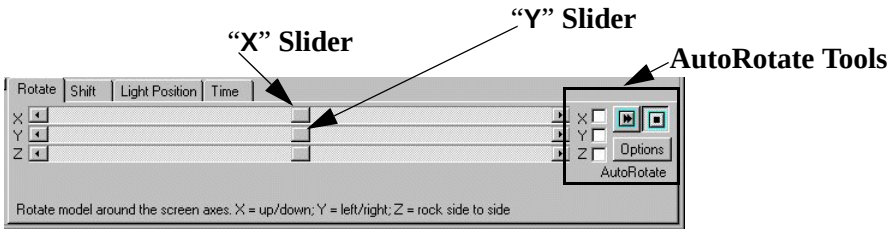
- ☞ **Default Settings** in the Model Tree panel
- ☞ **Background** in the Display Settings panel
- ☞ ☑ **Save configuration on close** to save these settings for future 3D-Explorer projects
- ☞ **[Apply]**

Section 20: Using the Navigation Tools

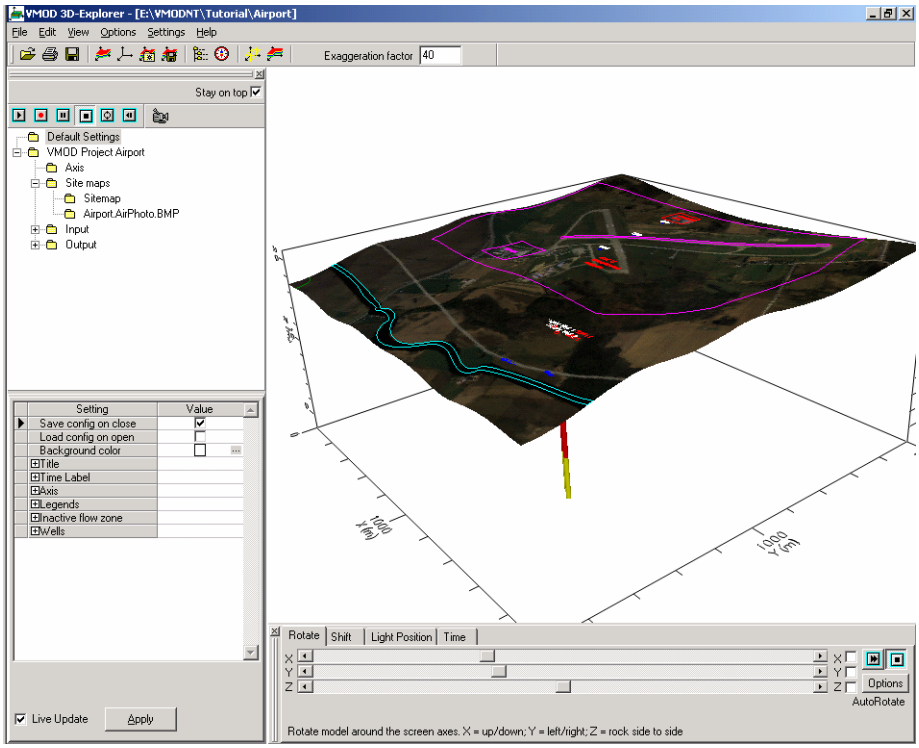
The **Navigation Panel** allows you to **Rotate** the model around the X, Y, and Z axes, **Shift** the display along the X, Y, and Z axes, and move the position of the **Light** source. For this example, you will rotate the model around the X-axis in order to gain a 3D perspective view of the model.

- ☞ **Drag** the "**X**" Slider to the left to rotate the top of the model "towards" you (as shown in the following figure)
- ☞ **Drag** the "**Y**" Slider to the left to rotate the right-hand side of the model "towards" you (as shown in the following figure)

☞ **Drag** the "**Z**" Slider to the left to "level the model (as shown in the following figure)



Your screen should now appear similar to the figure shown below.



NOTE: If your Airphoto map is not displayed, you may need to modify your display settings in **Settings>Open GL**.

Section 21: Creating a 3D Contaminant Plume

Next, you will create a 3D volumetric representation of the contaminant plume located beneath the Refueling Area by creating an isosurface for a selected concentration value.

The Airphoto sitemap obscures the view of the model below it. To remove the map from view

☞ ⊞ **Site maps** in the **Model Tree**

☞ ⊞ **Airport.AirPhoto.BMP**

☞ ☐ **Visible**

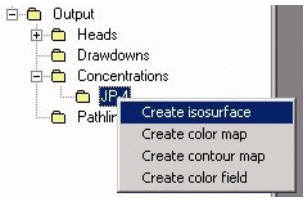
Now the model domain is clearly visible.

☞ ⊞ **Output** in the **Model Tree**

This will expand the **Output** branch and you will see a listing of the available Output data categories.

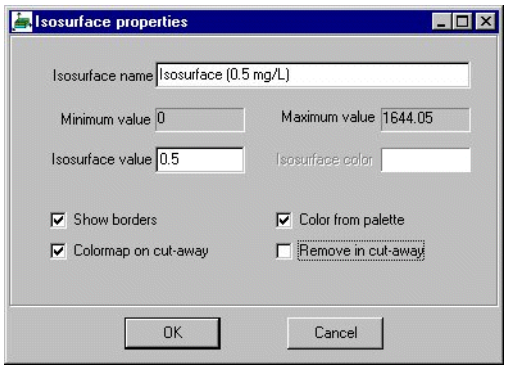
- ⊞ **Concentrations**

This will again expand the tree to include a listing of the available concentration species. The Airport4 model only simulated a single contaminant species with no degradation reactions, so there will only be a single entry (**JP-4**) under the **Concentrations** branch of the **Output** tree.



- **JP-4** to highlight it
- **Right-Click** on **JP-4** to see a popup list of display elements (as shown in the figure above)
- **Create isosurface**

An **Isosurfaces properties** window will appear, as shown in the following figure:



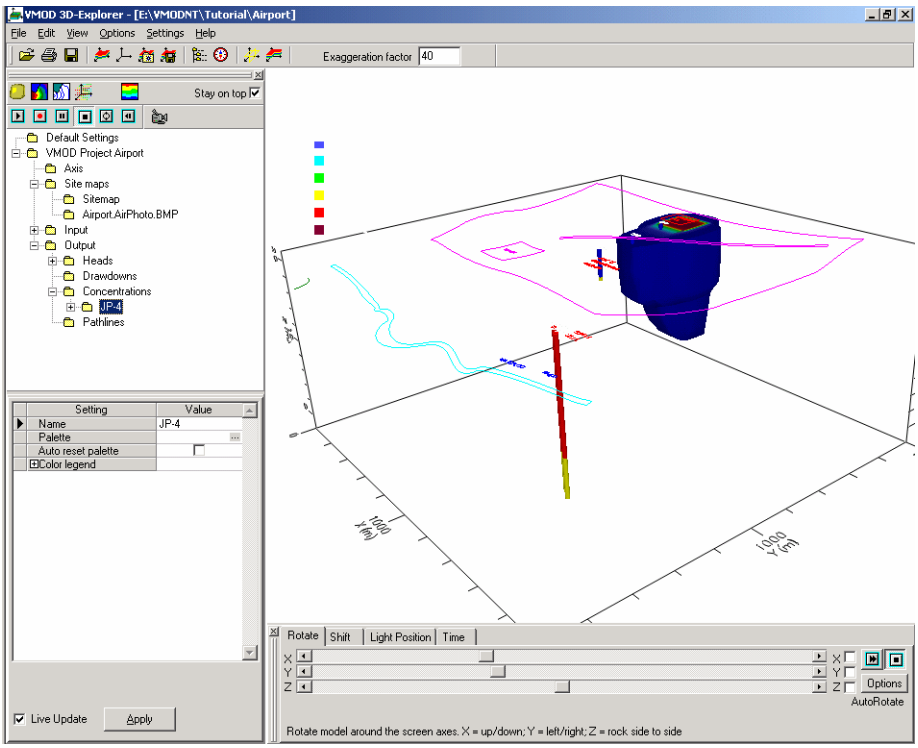
This window allows you to specify the concentration value, and the visual attributes for the isosurface. **Type** the following information:

Isosurface name: Isosurface (0.5 mg/L)

Isosurface value: 0.5

- **☑ Show borders**
- **☑ Colormap on cut-away**
- **☑ Color from palette**
- **☐ Remove in cut-away**
- **[OK]**

Your screen should now display an isosurface for a JP-4 concentration equal to 0.5 mg/L at the first output time step of 730 days (see figure below).



Notice that the concentration isosurface is clipped where the isosurface meets the water table. 3D-Explorer has built-in logic that removes concentration isosurfaces above the water table.

Customizing the Color Legend

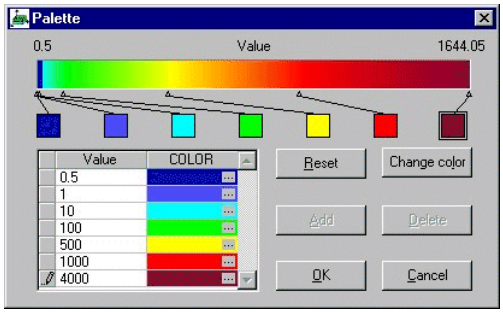
To change the text color for the Color Legend,

- ☞ **JP-4** in the **Model Tree** to highlight it
- ☞ ⊞ **Color legend** (in the Display Settings panel)

The Concentration values on the Color Legend should now be visible, but the default color scale should also be modified to improve the appearance of the Color Legend.

To customize the values associated with each color

- ☞ **JP-4** in the **Model Tree** to highlight it
- ☞ **Palette [...]**



A **Palette** window will appear as shown in the figure on right.

By default, the color palette for each data type is set to a seven color rainbow with equal value intervals between each color. This palette may be customized to any number of colors and any value associated with each color.

Type the color values shown in the above figure.

- ☞ **[OK]** to accept the new color palette

Selecting Output Times

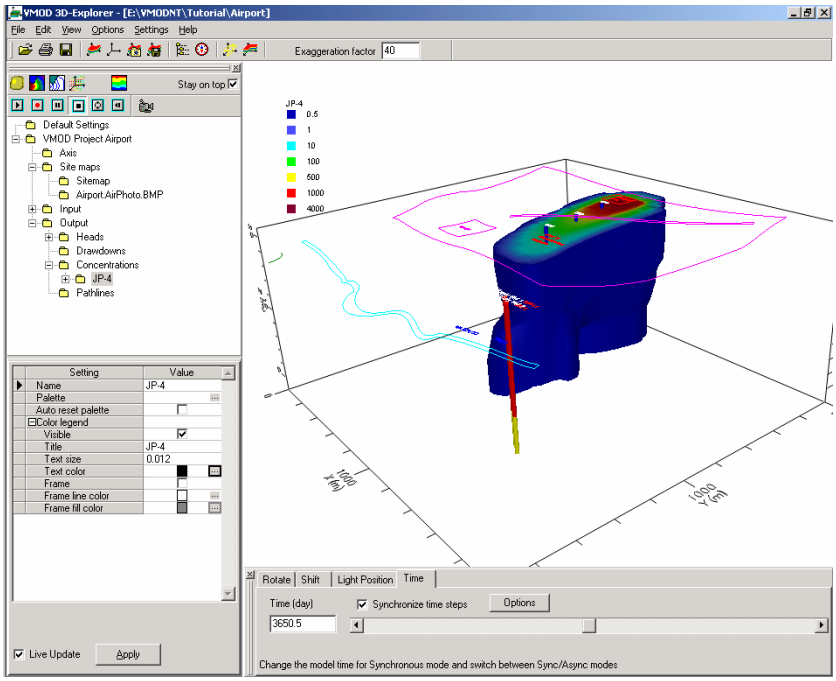
To examine the same isosurface value at a time of 3650 days,

☞ **JP-4** in the **Model Tree** to highlight it

In the **Navigation Tool Panel**, click on the **Time** tab. The available output times for the JP-4 concentrations can be displayed by using the slider button. These output times correspond to the output times specified in the Visual MODFLOW model. To select a different output time:

☞ **Click** in the **Time** value field and **Type: 3650** ↵

The screen will now display the results of the 0.5 mg/L concentration isosurface at 3650 days, and should appear similar to the figure shown below:



The Slider button can be used to view the isosurface for any of the available output times.

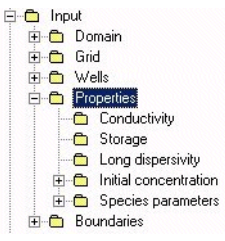
Section 22: Visualizing Model Input Data

Visual MODFLOW 3D-Explorer also allows you to display the model input data in 3D perspective.

☞ ⊞ **Input** in the **Model Tree Panel**

This will expand the Input branch to display the available input data categories which can be displayed using 3D-Explorer.

Now you will display the different Conductivity zones in the model.

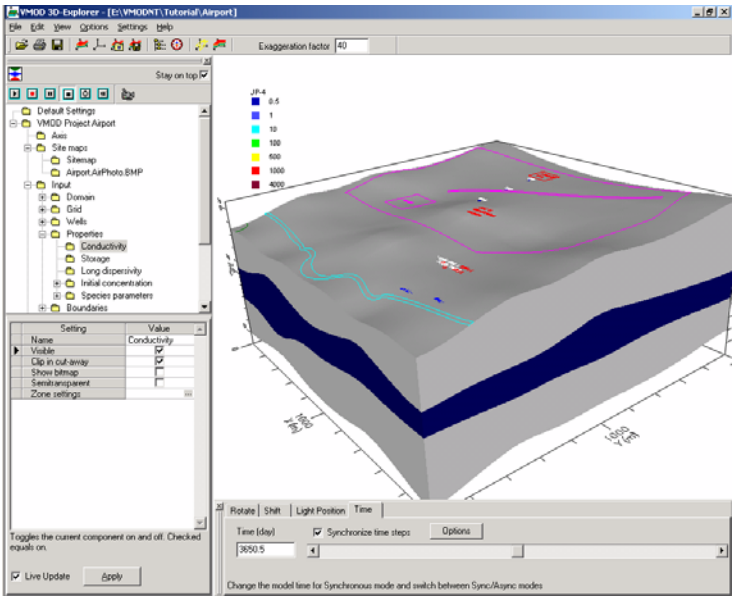


☞ ⊞ **Properties** in the **Model Tree**

☞ **Conductivity**

☞ ☑ **Visible** in the **Display Settings**

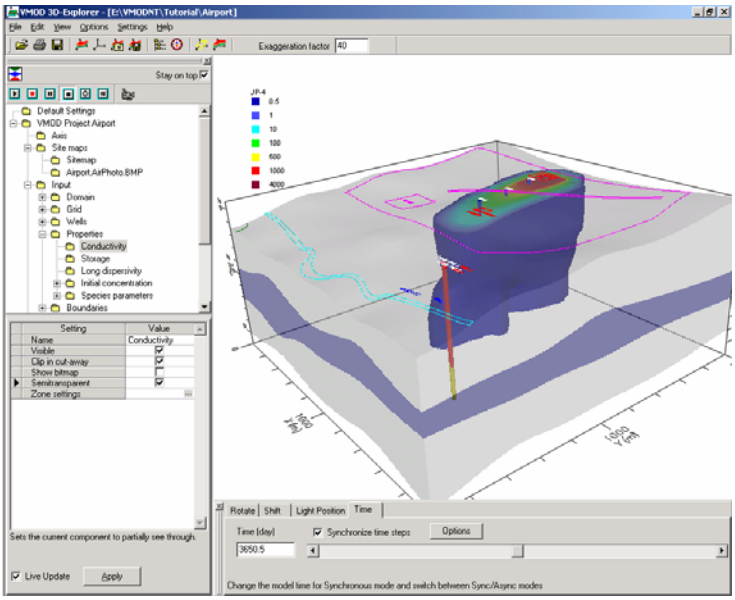
Your screen should now appear similar to the figure below:



Notice that the concentration isosurface is now “hidden” by the hydraulic conductivity regions. However, it is still possible to view the isosurface by changing the hydraulic conductivity from solid to semi-transparent.

☞ ☑ **Semitransparent** in the **Display Settings Panel**

The isosurface should now be visible through the hydraulic conductivity zones. The semi-transparent option is available for most input parameters, slices, and isosurfaces.



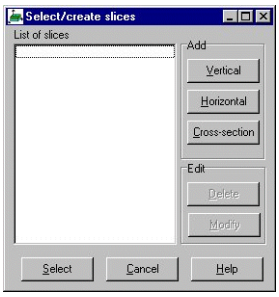
Section 23: Defining Surfaces (Cross-Sections and Slices)

3D-Explorer allows you to plot Color Maps and Contour Maps for selected model parameters on any horizontal, vertical, or cross-section surface through the model domain. However, first you must define the locations and orientations of the planar surfaces. The following section

describes the process of creating a cross-section, and a horizontal slice, through the model domain.

- ☞ **Options/Define slices** from the top menu bar

The **Select/create slices** window will appear, as shown in the figure on right. This window contains a list of all the horizontal, vertical, and/or cross-section slices defined within the model domain. Currently, the list is blank because you have not yet defined any slices.

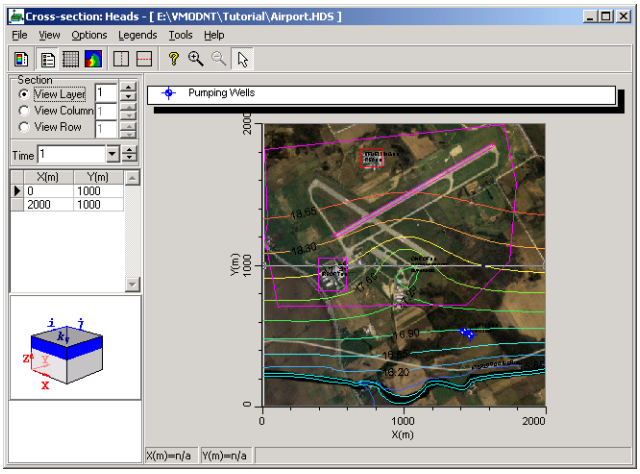


First, you will create a cross-section through the model.

- ☞ **[Cross-section]** button

This will open the **Cross-section** window, as shown below.

The default location for the cross-section is a horizontal straight line from the left side to the right side of model domain. In this example you will modify the default cross-section line such that it cuts along a path from the North Central region to the South Eastern region of the model.



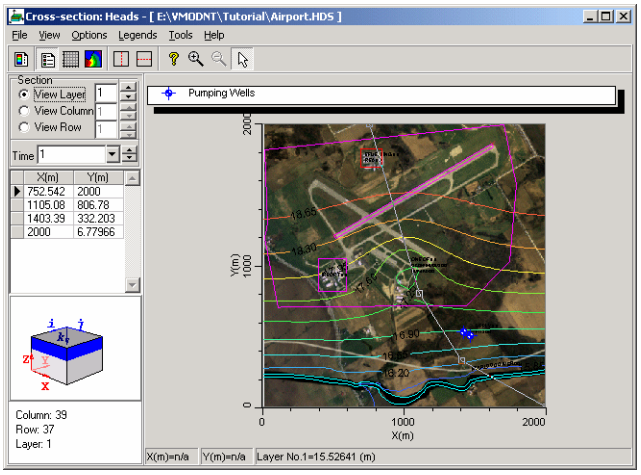
In some 3D visualization packages, cross-sections are required to be straight lines through the model domain. With 3D-Explorer, cross-sections can be defined along any polyline through the model domain. These polylines are defined by a series of **vertices.** By default, the cross-section starts with just two vertices (a straight line), but new vertices can be easily added by **Right-Clicking** on the cross-section line at the desired location of the new vertex.

To modify the existing cross-section,

- ☞ **Drag** the left vertex to a point just North of the Refueling Area
- ☞ **Drag** the right vertex to the southeast corner of the model domain
- ☞ **Right-click** on the cross-section line and select **Add vertex**

A new vertex will appear on the line. This vertex can be dragged to any location on the map.

- ☞ **Drag** the new vertex to a location just southwest of the region where the aquitard pinches out (as shown on the figure below)
- ☞ **Right-click** on the cross-section line again and select **Add vertex**
- ☞ **Drag** the new vertex to a location between the second vertex, and the southeast corner of the model, as shown on the figure below



Once you have defined the cross-section line similar to the figure shown above,

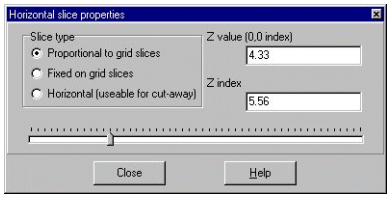
☞ **File/Close** from the top toolbar

The cross-section (**Cross-section no 1**) you just defined now appears in the **Select/create slices** window.

Next you will create a horizontal slice through the model domain.

☞ **[Horizontal]**

This will display the **Horizontal slice properties** window, as shown in the figure to the right:



The slider button can be used to move the horizontal slice to any vertical location in the model domain.

☞ **[Close]** to accept the default values

This Horizontal Slice (**Horizontal slice no 1**) will also now appear in the **List of slices**.

You can define as many different cross-sections, and horizontal and vertical slices as you like throughout the model domain. These slices can be used to define cut-away regions, or to plot Color Maps and Contour Maps of selected model properties and simulation results (e.g. Conductivity values, heads, concentrations).

For this example you will only define these two slices.

☞ **[Select]** to close the Select/create slices window.

Section 24: Creating a Cut-Away View

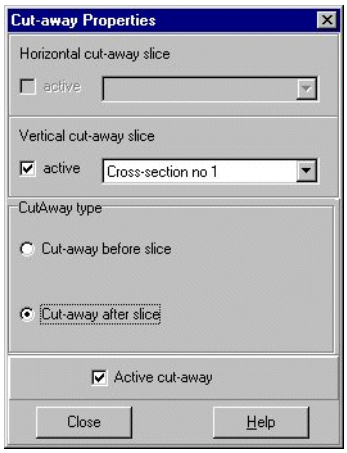
Next you will create a cut-away view of the model.

☞ **Options/Select cut-away** from the top menu bar

This will open the **Cut-away properties** window, prompting you to select the surface you would like to use to define the cut-away region. For this example you will select the cross-section you defined earlier:

To select the cross-section,

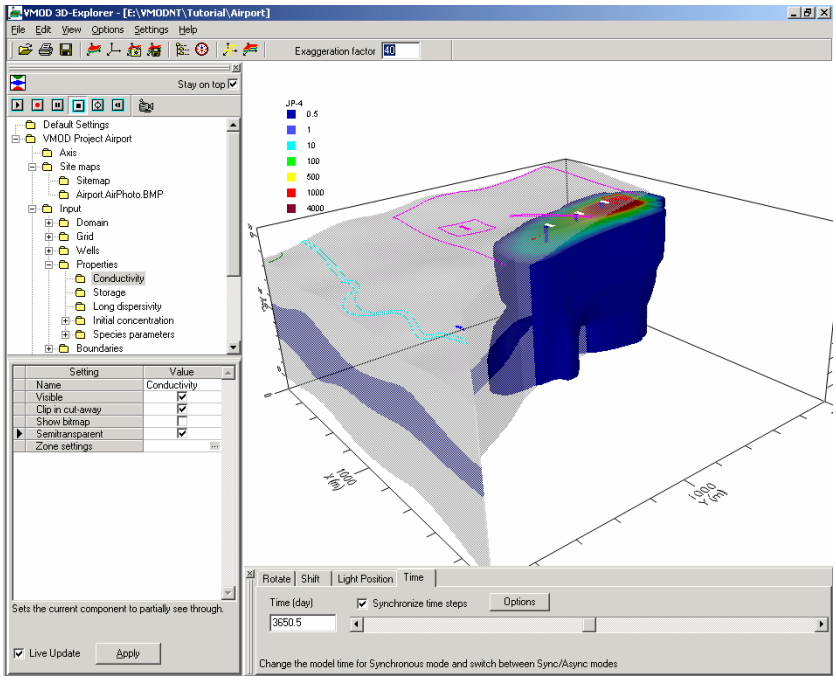
☞ to open the **Vertical cut-away slice** pull-down menu, and select **Cross-section no. 1** (as shown below).



☞ **⊙ Cut-away after slice**

☞ **[Close]**

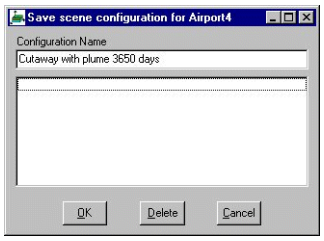
The model display should appear similar to the figure below. Notice that the conductivity zones are removed from the cut-away region, while the concentration isosurface remains. This is possible because each displayed object has it's own "Remove from Cut-away" setting. In the case of the concentration isosurface, you de-activated this option. Therefore, the concentration isosurface is NOT removed in the cut-away region.



This view of the model data can now be saved for easy retrieval at a later time.

☞ **File/Save scene configuration** from the top menu bar

The **Save scene configuration** window will appear, as shown below:



Using this window, the scene configuration can be saved with a user defined name, and this configuration can be recalled every time the same project is opened.

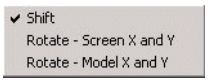
Type a **Configuration Name** of **Cutaway with plume at 3650 days**.

☞ **[OK]**

If the most recently viewed scene configuration is saved using the name "**Default"**, VMOD 3D-Explorer re-opens the project automatically with the "**Default**" saved settings.

To examine the model view from different angles, the sliders on the **Navigation Panel** can be used to rotate the view, or you can move and rotate the view as follows:

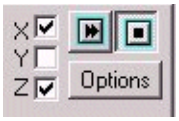
Right-Click anywhere in the model screen



☞ **Rotate - Screen X and Y**

With the left mouse button, click and drag anywhere in the model screen. Try to achieve several different views of the model and cut-away isosurface.

The **AutoRotate** tools located on the right-hand side of the Rotate tab may be used to continuously rotate the model image around the X, Y, and Z axes. By checking off one or more axes, then using the **Play** and **Stop** buttons, you can start and stop the rotation of the image. The **Options** button allows you to change the speed of rotation



When you have finished rotating the model, return to the previously saved scene configuration.

☞ **File/Load scene configuration** from the top menu bar

This will produce the **Load scene configuration** window:

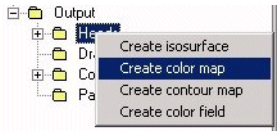
☞ **Cutaway with plume at 3650 days**

☞ **[OK]**

Your screen will now be returned to the previously saved view.

Section 25: Creating a Color Map of Head Values

Next you will create a colormap of the calculated heads plotted on the previously defined horizontal surface.



☞ ⊞ **Output** in the **Model Tree**

☞ **Heads** in the **Model Tree** to highlight it

☞ **Right-Click** on **Heads** and select **Create color map** from the popup menu.

The **Select/create slices** window will appear, prompting you to select one of the available slices where the color map will be plotted.

☞ **Horizontal slice no. 1** to highlight it

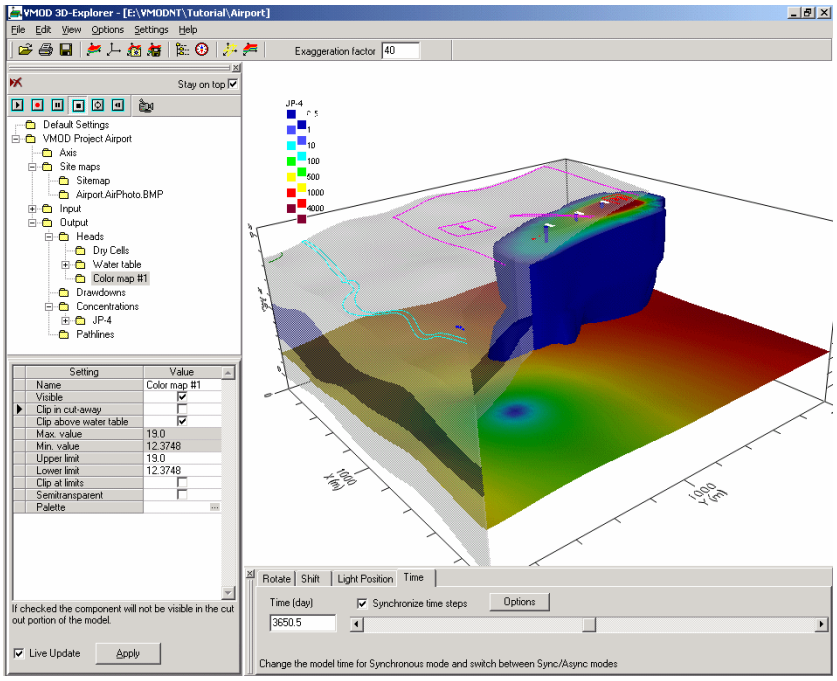
☞ **[Select]**

A Color Map of the Heads distribution on the selected horizontal slice should now be visible. However, it is not displayed in the cut-away region of the model.

To plot the Heads Colormap in the cut-away region of the model:

- ⊞ **Heads** to expand the Heads branch
- **Color map #1** under the Heads branch
- ☐ **Clip in cut-away** in the **Display Settings Panel**

The Heads Colormap should now appear in the cut-away region of the model, as shown in the following figure.



Customizing the Color Legend

As you may notice, there are now two Color Legends appearing on the screen. These Color Legends can be moved to any location in the 3D Display window to enhance the appearance of the image.

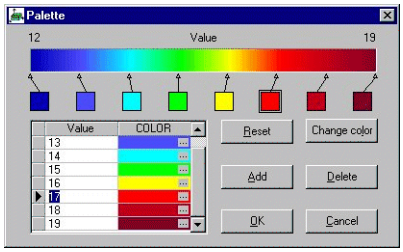
To move the Heads Color Legend, click-and-drag the legend to the right-hand side of the Display Area.

Adding a New Color

To modify the color palette for the Heads Color Map,

- **Heads** to highlight this data type (under the Output data type)
- **Pallete [...]** button in the **Display Settings Panel**

The **Palette** window appears.



Change the default color **Value** settings to 12, 13, 14, 15, 16, 17, and 19.

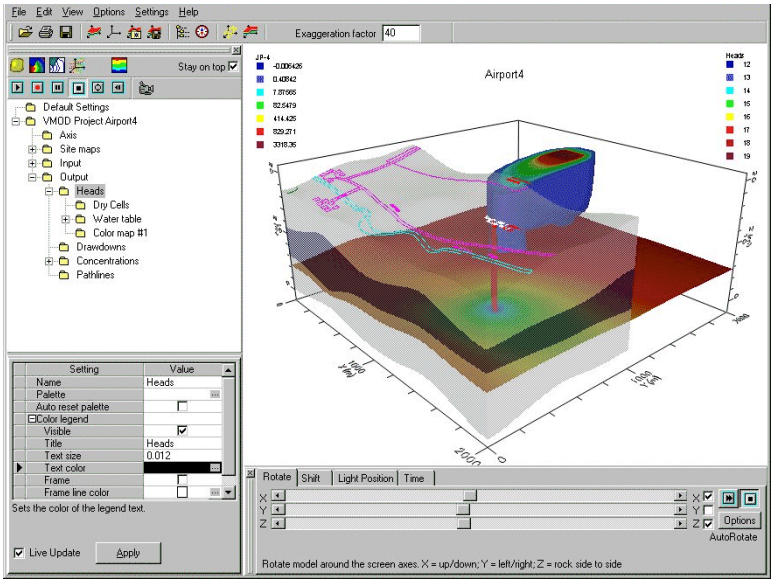
- ☞ **Click** in the **Value** field on **17**
- ☞ **[Add]** button (to add another color value)

A new color will be added to the palette for a **Value** of **18**.

The **Palette** window will appear as shown in the figure on the right.

- ☞ **[OK]**

The Heads Colormap legend should appear similar to that shown in the following figure:



Section 26: Printing to a Graphics File

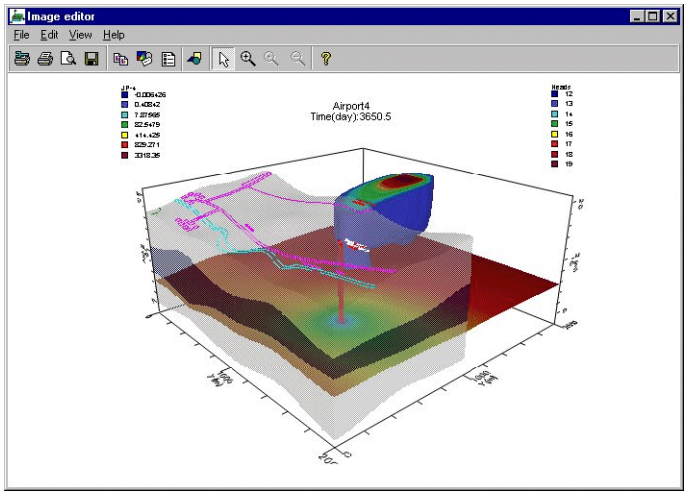
At this point it may be desirable to generate a hardcopy print of the 3D model image to include in a report.

- ☞ **File/Print** from the top menu bar

A high-resolution bitmap image will be created of the on-screen image, and displayed in an **Image editor** window as shown in the following figure.

The **Image editor** window is used to preview the image that will be printed, and to add text, lines, shapes, and titles to the image.

The Image editor window can also be used to customize the **Page setup** to include a detailed description of the image, a company logo, and a project name.



Once the image and Page setup are completed,

☞ **File/Print** from the top menu bar of the **Image editor** window.

Alternately, the image can be saved to a graphics file in bitmap (.BMP), Windows Metafile (.WMF), Enhanced Windows Metafile (.EMF), .JPG, .TIF, .GIF, or .PNG file formats.

☞ **File/Save as graphics file...** from the top menu bar of the **Image editor** window

A **Save to file** window will appear where you can select the file type and specify the file name and location.

☞ **File>Exit** to close 3D Explorer.

This concludes the step-by-step Demo tutorial for Visual MODFLOW and Visual MODFLOW 3D-Explorer.